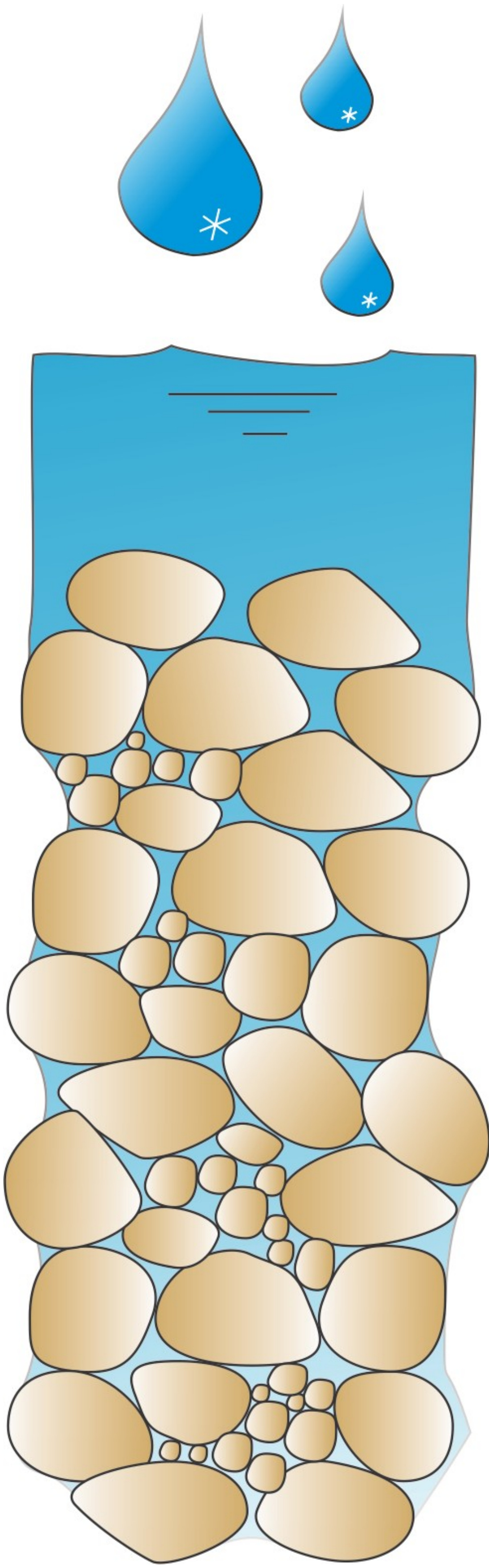


MELEF FreshSaltWatershed Help Manual

Help Manual of MELEF FSW model includes an introduction to the scope and capabilities of this free software, a user manual guide with a complete description of the software windows, a technical manual with the description of the principal equations and methodologies implemented, and a step-by-step tutorial of application case to the flooding of the Meirama Open Pit Mine.

Dr. Jesús Horacio Hernández Anguiano
Dr. Francisco Padilla Benítez

Guanajuato, México. 2022

Univerisdad de Guanajuato México
Universidad de La Coruña, España.

HELP MANUAL OF MELEF FSW

GIS Toolbox - User Interface - Numerical Model.

by MELEF FSW

Products MELEF FSW.
Developers
Dr. Jesús Horacio Hernández Anguiano
Dr. Francisco Padilla Benítez

Guanajuato, México. 2022

MELEF FreshSaltWatershed Help Manual

Printed: May 2022 in Guanajuato México.

Editors

Dr. Jesús Horacio Hernández Anguiano
Dr. Francisco Padilla Benítez

Un agradacimiento especial para:

The researchers of University of A Coruña who worthly contributed to the development of the code FreshWaterSheds. Likewise, were very useful, the financial support of Ministry of Education and Science CICyT (CGL2006-01452 y CGL2009-11258), the postgrade grant of Xunta de Galicia (Program María Barbeito), the involvement, support and data supply of LIMEISA mining company, as well as the European Science Foundation

Contents

Foreword 9

Part I INTRODUCTION TO MELEF FSW 11

1 Required software 14
2 Installation and configuration 16
3 Scientific publications 20

Part II USER MANUAL 23

1 Global steps and usable information 24
2 User interface 25
Create or open a simulation project 25
Toolbar description 27
Panel 1: Simulation control 27
Panel 2: Simulation conditions 30
Panel 3: Parameters / resolution 35
Menus and Tools 41
Menu Project 41
Menu Mesh 42
Process Mesh (geo-2dm-msh) 42
Process IWFM Mesh (dat) 43
Menu Properties 43
Create SEC 43
Create PRN 44
Create SOI 46
Create SLR/CND/SLC 49
Transient Database 52
Menu Utilities 53
Open simulation folder 53
Menu Results 53
Global ETA 53
Flows DEB 59
Nodes VNO-SOL-VEL 63
Zonal SOL 69
Animations 1D VNO-VEL 73
Animations 2D VNO-SOL-VEL 76
Zonal Balance 80
3 Toolbox GIS 85
1. Import Mesh 85
1.1 Import Nodes - Mesh (.GEO - .2DM - .MSH) 85
2. Non Saturated Zone - EvapoTranspiration (SOI) 87
2.1 New Non Saturated Zone 87
2.2 New Ksuperficial 89
2.3 Generate SOIMELEF 90
3. Geological Materials - Impervious Substratum (PRN) 90
3.1 Approximate an Impervious Substratum 90
3.2 New Geology 93
3.3 Generate PRNMELEF 94

4. Boundary and Inner Conditions (CND - SLC) **95**
4.1 New CND - SLC 95
4.2 Generate CNDMELEF or SLCMELEF 96
5. Rainfall and Water Uses (SLR) **97**
5.1 New Rainfall - Water Uses 97
5.2 Generate SLRMELEF 99
6. Gauging Sections (SEC) **100**
6.1 New Gauge Section 100
6.2 Generate SECMELEF 101
Utilities (ArcMAP ESRI) **101**
Adjust Infiltration Rate 101
Adjust KZ by Slope 102
Adjust Soil Thickness 102
Adjust ES by Slope and Vegetation 102
Generate Simulation Files 104
Generate simulation file COR 104
Generate simulation file INI 105
Reconditioning DEM and MESH 106
DEM Burning by Slope 106
DEM Burning Elevation 108
DEM Reconditioning Multiple 109
Fill Sinks in DEM 111
Generate Buffers to Create Mesh Zones 112
Redistribute Vertex 113
Topographic Depressions Evaluation 114
Subfunctions 115
Delete Extra Fields 115
Update Field by Raster 116

Part III TECHNICAL MANUAL 119

1 MELEF FSW 120

Input simulation files **120**
Input COR file 120
Input CDN file 120
Input ELE file 121
Input INI file 121
Input INP file 122
Input PRN file 124
Input SEC file 125
Input SOI file 125
Input SLC file 126
Input SLR file 127
Result simulation files **130**
Output DEB file 130
Output ETA file 132
Output FIN file 137
Output SOL file 138
Output VEL file 139
Output VNO file 139
Formulations ground/surface models **140**
Groundwater Model 141
Surface Model 143
Reservoir operations **150**

Floodgates simulation ... 150
Spillways simulation ... 152
Interaction models ... 156
Continuous evaporation and transpiration model ... 156
Discontinuous EvapoTranspiration model ... 158
Overland flow model ... 159
Soil water balance and groundwater net recharge ... 159
Numerical conditions and resolution ... 160
Tunnels and Galleries simulation ... 161
Automatic water bypass ... 163
Rules of convergence/calibration ... 168
2 USER INTERFACE ... 172
Units of measure ... 172
Excel/CSV standard date formats ... 173
Interpolation as histogram with backward step. ... 174
Multiple file selection window ... 175
Video Recording ... 176
3 GIS TOOLBOX ... 177
Reconditioning of a raster elevation model ... 177
Code of use ... 181
Part IV APPLICATION AND TRAINING 183
1 Practical case 1: Flooding of Meirama open pit mine (River Mero Basin) ... 184
Introduction ... 184
Objectives ... 191
Practice 1: Preparing the numerical model: ... 191
Preparing the Simulation conditions ... 191
Preparing the Parameters/resolution ... 197
Preparing the transient database .XLS ... 200
Practice 2: Preparing GIS simulation layers ... 204
Importing the mesh to GIS ... 204
Create SOIMELEF with soi properties and EvapoTranspiration rates ... 205
Create PRNEMELF with geological material properties ... 206
Create CNDMELEF/SLCMELEF with boundary conditions ... 207
Create SLRMELEF with the rainfall and water uses ... 207
Create SECMELEF with the gauging sections ... 208
Practice 3: Preparing and running MELEF FSW ... 208
2 Activate Fresh/Salt Water Model ... 210
Índice 219

Foreword

The constant development of MELEF FreshSaltWatershed numerical model and its capabilities allow the generation of new numerical solutions that approximate the hydrological reality of continental and coastal basins.

All this requires the constant evolution of the spatial information management through the Python scripts developed to create the GIS Toolbox, and the Graphical User Interface that manage the pre and post-processing of the simulation conditions and results.

Finally, this second version of the Help Manual describes the scope, capabilities, operation and characteristics of the FreshSaltWatershed Management System.

MELEF FSW HELP MANUAL

Numerical Hydrological Model for
the integrated simulation of regional
groundwater and surfacewate flows
in continental and coastal
watersheds.

Part I

1 INTRODUCTION TO MELEF FSW

In distributed hydrological modeling, there are commercial reference codes such as Mike She, Mike Basin, Cathy, ModFlow, Feflow among some other models. The development of physical based distributed numerical technologies, which are an alternative to other commercial solutions, are usually not relevant when they do not provide confident solutions to most hydrological processes. In this context, researchers from the University of La Coruña and the University of Guanajuato join forces to develop a numerical solution with a series of qualities and capabilities that distinguish it from other codes. This approach allows solving the groundwater and surfacewater flows of a hydrological region in an integrated way for most hydrological processes. The FreshSaltWatershed (FSW) model, as one of MELEF models (Modèles d'ÉLÉments Finis, for its acronym in French), considers novel capacities to solve the continental and coastal regional groundwater and surfacewater flows in an integrated way, as most regulation problems and seawater intrusion through an immiscible interface of fresh-salt water as it is represented in Figure 1.

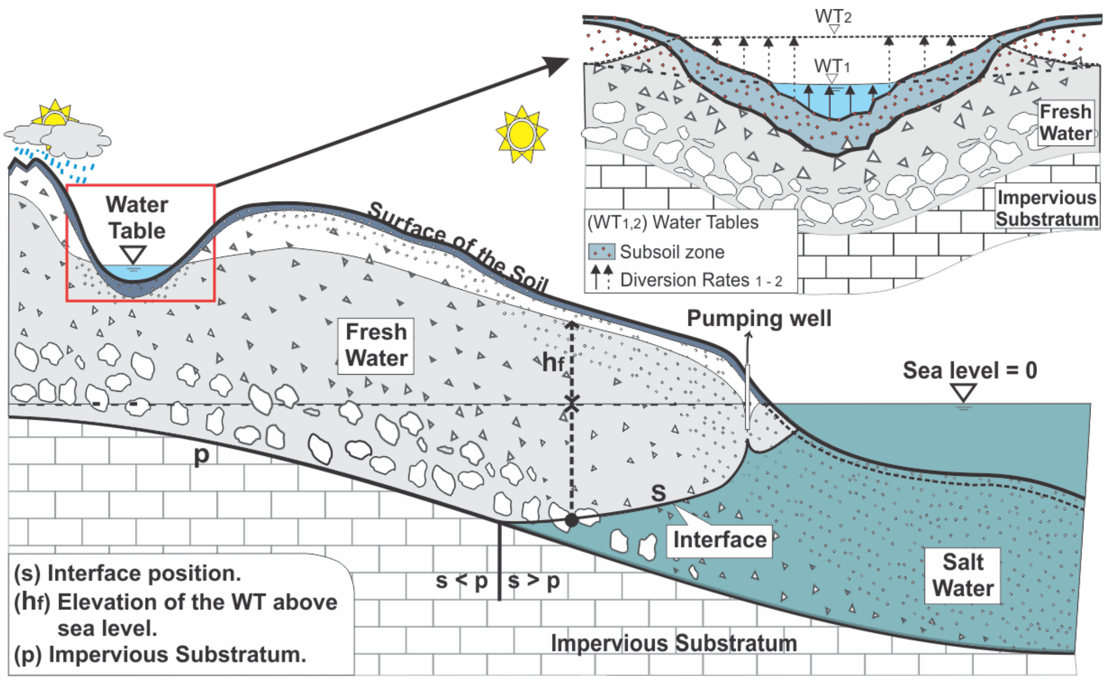


Figure 1: Hydrologic representation of groundwater and surface, continental and coastal water flows, solved by the MELEF FSW model.

In addition, the MELEF FSW model implements a User Interface developed in Matlab that manages pre- and post-processes concerning the input information and the results generated by the model. Some of the principal results analysis capabilities are presented in Figure 2. These allow to analyze results meanwhile the model is running, create figures 1D or 2D to represent the information and take decisions during the calibration process or generate figures for publication purposes.

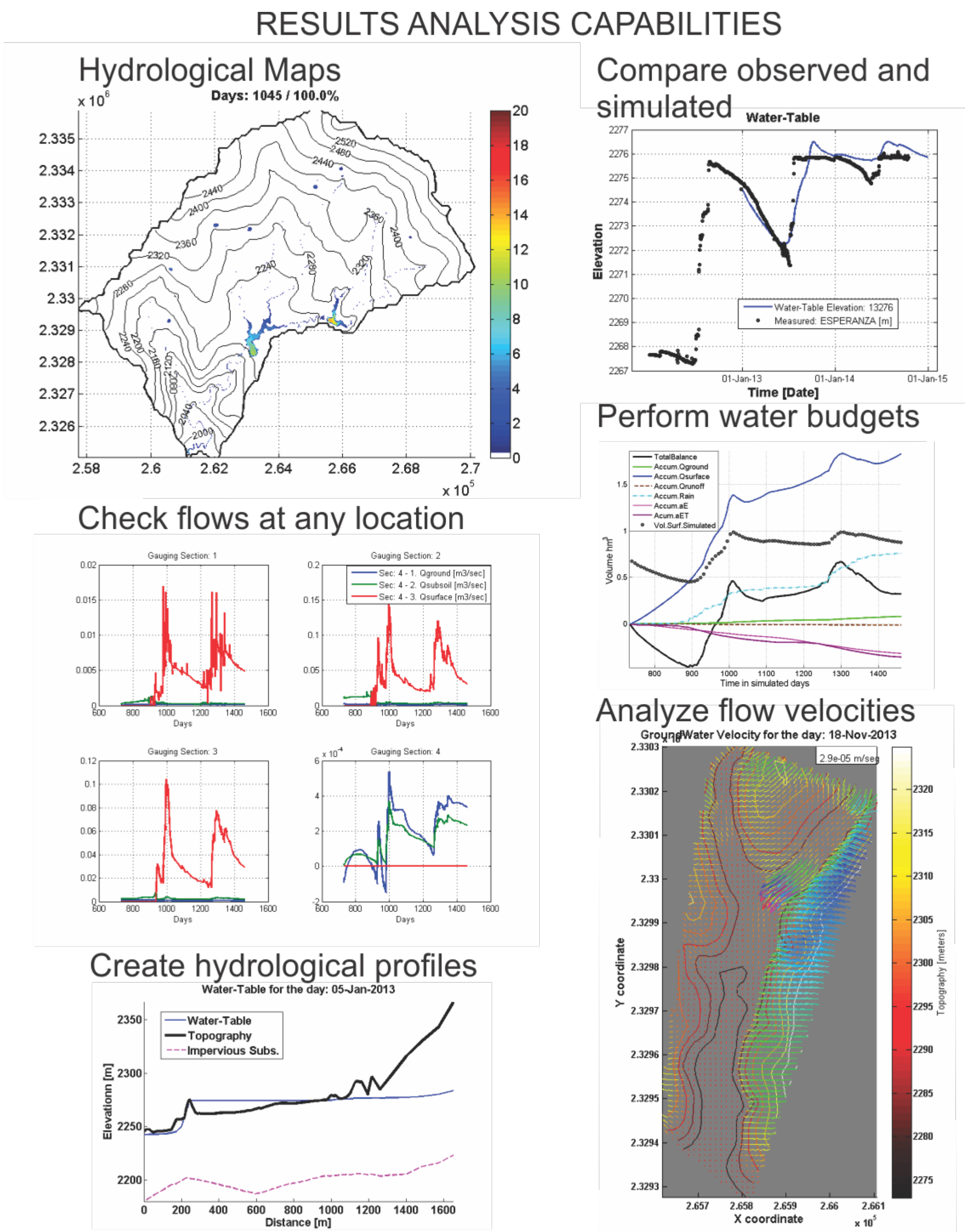


Figure 2: Results analysis capabilities of MELEF FSW User Interface.

Today Geographic Information Systems (GIS) are an essential tool for managing any type of information with spatial distribution and have the capabilities to manage the information needed by this kind of numerical models, which have also the clear advantage of counting on the analysis power of GIS. Therefore, the MELEF FSW Toolbox was developed for ESRI ArcGIS and QGIS platforms. The MELEF FSW Toolbox is made up of six tool sets to manage the information into simulation conditions, see the Figure 3.

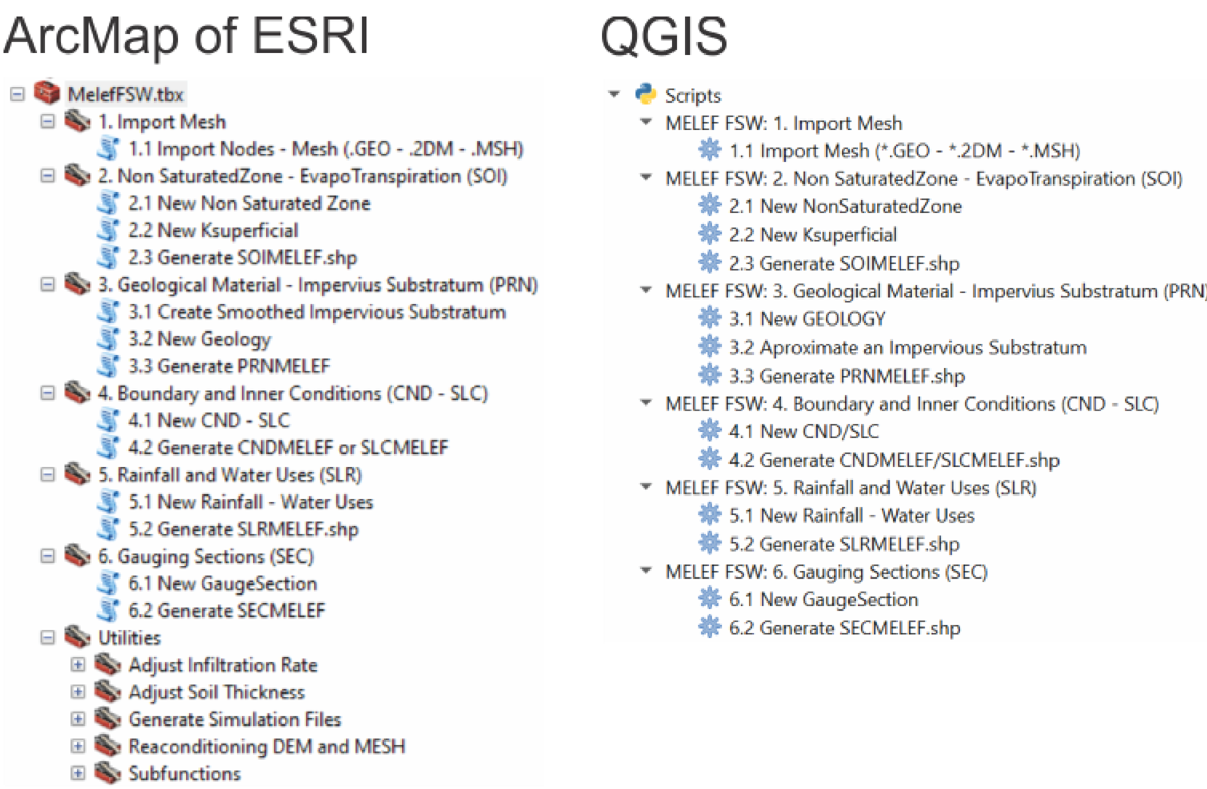


Figure 3: Toolbox MELEF FSW versions for QGIS and ArcMap of ESRI.

1.1 Required software

The preparation of a new Project of simulation with MELEF FSW requires the generation of the triangular element mesh, the management of the simulation and boundary conditions, the storage of historical data and measured values, to prepare the simulation files to run the model, and to analyze results with figures and perform water budgets automatically. The software required and its function is depicted in the diagram in Figure 4.

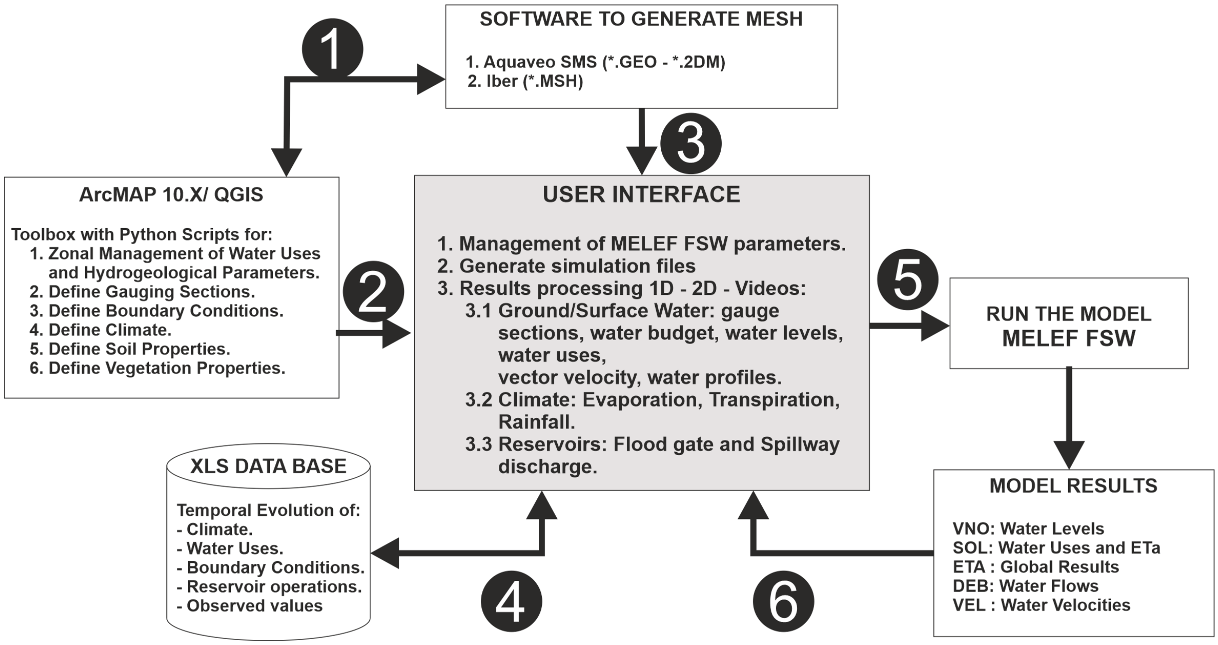


Figure 4: Diagram with the software required for management of a new Project.

Description of the software required to perform the following tasks:

1 **Creation of the triangular finite element Mesh**: The software to create the mesh could be Aquaveo Surface Water Modeling System (Aquaveo SMS) or free software like IBER 2D Hydraulic Modeling or even the IWFM Mesh Generator. These programs create the mesh in different formats that you can use to import it into the GIS.

Use the MELEF FSW Toolbox to Import the mesh and create the required shapefiles to start: The MELEF FSW Toolbox is fundamental to create shapefiles with the correct attribute table structure and to process the required information such as number of nodes, area of influence of nodes, among other information extracted from a Digital Elevation Model. There are two versions of MELEF FSW Toolbox, the first version was designed to run on ESRI's ArcMAP platform and the other to run on the free software QGIS.

2 **Create the simulation files INI, CND, PRN, SEC, SLR, SOI, INP required to run the MELEF FSW model:** The User Interface translates the simulation and boundary conditions layers into these simulation files, the required information are the layers generated with the Toolbox MELEF FSW for GIS.

3 **Create the COR and ELE files with the geometry of the mesh:** The MELEF FSW User Interface translate the mesh files (*.GEO; *.2DM; *.MSH) to generate the files *.COR (coordinates XY) and *.ELE (triangular elements) required by the MELEF FSW model.

4 **Create transient simulation files with historical records:** Implement an Excel file (*.XLS, *.XLSX) to store the transient behavior of water extractions/injections, water management, reservoir operations, topography modifications, boundary conditions, climate and store observed values.

5 **Run the MELEF FSW numerical model:** The user Interface check all the required files and prepares the folder to run the numerical model. The MELEF FSW numerical model was programed in Fortran 77, it runs in any windows platform and requires minimum memory to solve meshes with better less than 1 million nodes.

6 **Create Figures, Maps, Profiles, Water Budgets and Videos:** The User Interface can analyze results in the output files of the MELEF FSW model. The analysis of results can be performed during or after running the numerical model.

1.2 Installation and configuration

The MELEF FreshSaltWatershed (FSW) User Interface was created as stand-alone executable of Matlab R2012a (7.17, 32 bits) for Windows. As it is a stand-alone executable, it allows running the program through the Matlab Runtime R2012a, version 7.17 with 32 bits. The Matlab Runtime can be downloaded for free from the following link: https://la.mathworks.com/products/compiler/matlab-runtime.html.

Matlab Runtinme contains the minimum libraries necessary for executables to work. If you already have Matlab installed, or the Matlab Runtime, in version R2023b (23.2, 64 bits) for Windows, it is not necessary to install the Runtime as well/again. If required, download and install the Runtime in the correct version, the User Interface will not run without the right version of Matlab / Runtime installed.

The next step is to execute the MELEF FSW.exe package to install the User Interface, the GIS Toolbox and Python Scripts, the MELEF FSW Numerical Model, and Manuals in PDF and CHM formats.

Respect to the GIS Toolbox for QGIS and ArcGIS (ESRI) platforms, it is developed with Python Scripts. For ArcMAP the GIS Toolbox works on versions 10.1 to 10.8.2. For QGIS the Python Scripts were developed in version 3.10 (A Coruña) and higher versions of QGIS would work with the MELEF FSW Toolbox.

Regarding the operating system and the characteristics of the processing equipment, it should be considered that ArcGIS or QGIS are required for management and transformation of spatial information that demands specific characteristics for graphic design. Whereas the User Interface through the Matlab Runtime can be less demanding of CPU processing or graphics, the calculation time of MELEF FSW model will depends directly on the processor capacity. Therefore, the following minimum characteristics are suggested:

OPERATING SYSTEM	Windows 32/64 bits XP, Server, 7, 8, 10
PROCESSOR	Intel Core 2 duo or higher (equivalent platforms) with at least 2 GHz of clock speed.
RAM	> 8 GB
DISK STORAGE CAPACITY	> 100 GB of free storage capacity.
GRAPHIC CARD	> 64 MB (required for ArcGIS or QGIS geoprocessing tasks)

Table 1: Table with suggested minimum CPU characteristics

Installation Process

User Interface

STEP 1: Verify that you have installed the Matlab Runtime or the Matlab software in the version 2023b 23.2 64 bits. If you already had a working version of the user interface, then it is no longer necessary to reinstall Runtime / Matlab. So far, all versions of the user interface have been compiled in Matlab version R2023b (23.2, 64 bits).

STEP 2: It is recommended to start the user interface installation using administrator permissions. To do this, right click on the executable package and select run as administrator. This prevents known problems of access permissions and editing databases managed by the program during operation. In versions of Windows 7 and higher, it may be necessary to go to the task scheduler to run the interface as administrator.

STEP 3: When starting the user interface, it will take a moment to display, because the Matlab Runtime must first be loaded and then the user interface is executed.

Open the GIS Toolbox in ArcMAP

Use the following steps to open the Toolbox MELEF FSW in ArcMAP.

STEP 1: Open ArcMAP and go to the menu Windows>Catalog and open the Catalog window.

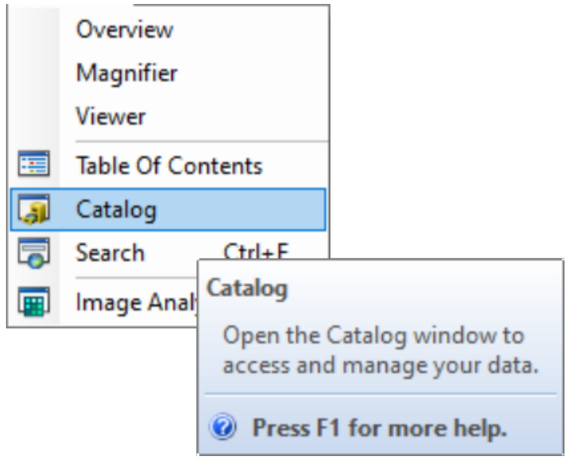


Figure 5: Menu "Windows>Catalog" to open the catalog window in ArcMap.

STEP 2: In the Catalog window click the button to add the installation folder of MELEF FSW.

STEP 3: Open Folder Connections in Catalog window and search for MELEF FSW.tbx file and open it, see Figure 6

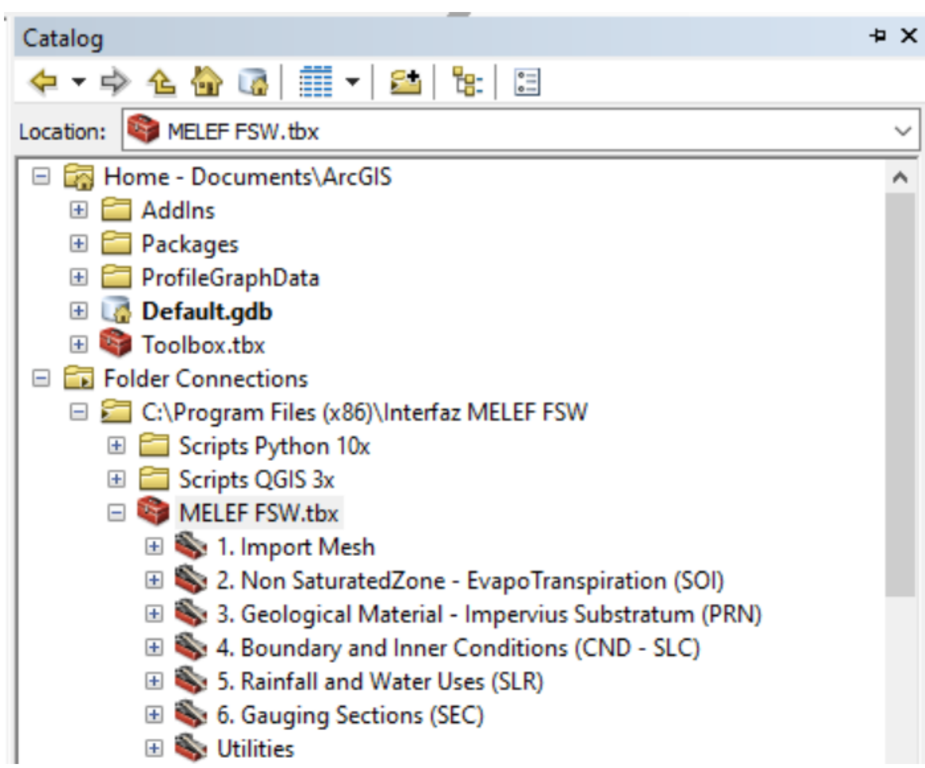


Figure 6: Path to the Toolbox MELEF for ArcMap of ESRI.

STEP 4: Open the tool sets to find the tool required to execute a task.

Open the GIS Toolbox in QGIS

Follow the process shown in the Figure 7

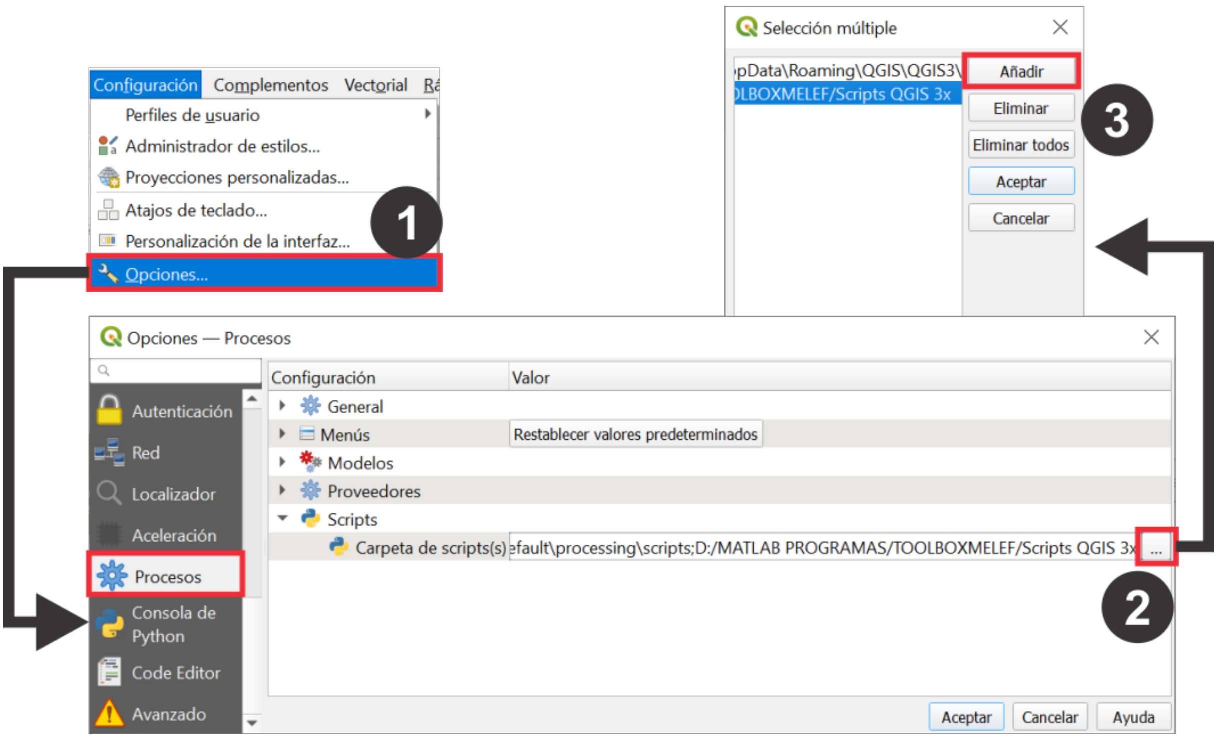


Figure 7: Steps to add the GIS Toolbox to QGIS.

1 : Open QGIS and go to the menu Configuration>Options..., in the Options window go to Processes.

2 : Click over Scripts to expand it and double click in Folder Scripts path to activate the button ... and click on it to open the window Multiple Selection.

3 : Add the path to the folder Scripts QGIS 3x that is in the installation folder. After this, the GIS Toolbox appears in the Scripts section of the QGIS Processing Toolbox.

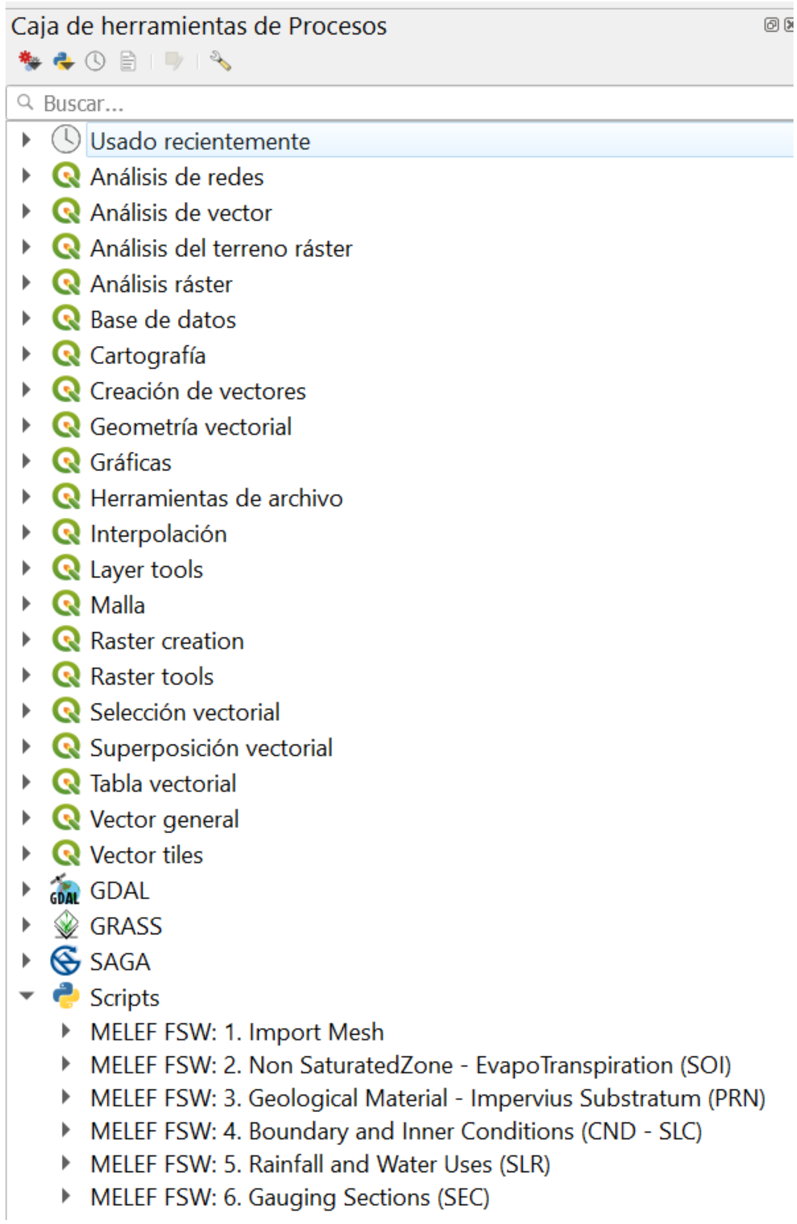


Figure 8: GIS Toolbox for QGIS after import the folder "Scripts QGIS 3x"

1.3 Scientific publications

Links to publications related to the application of the MELEF FSW numerical model in water resources management under different environmental conditions. These applications were made possible through research projects with various European funding sources.

- 2008. Numerical modelling of surfacewater/groundwater flows for freshwater/saltwater hydrology: the case of the alluvial coastal aquifer of the Low Guadalhorce River, Malaga, Spain. Environmental Geology.
- 2012. A numerical solution to integrated water flows: Application to the flooding of an open pit mine at the Barcés river catchment – La Coruña, Spain. Journal of Hydrology.
- 2015. A numerical solution for the integrated analysis of water resources management - Application to the Mero River Watershed - La Coruña Spain. Journal of Water Resource and Protection.
- 2015. Application of a numerical model designed for integrated watershed management. WIT Transactions on Ecology and The Environment.
- 2016. Improvements in Mero River Basin Water Supply Regulation Through Integration of a Mining Pit Lake as a Water Supply Source. Mine Water and the Environment.
- 2016. Modelling Integrated Extreme Hydrology. International Journal of Safety and Security Engineering.

Part II

2 USER MANUAL

2.1 Global steps and usable information

Globally, the most relevant steps and the information that can be used in a new simulation project with MELEF FSW are shown in Figure 9. It is important to clarify that not all the listed information is necessary to execute a simulation project, this will depend on the hydrological conditions and water uses involved in the environmental problem to be simulated.

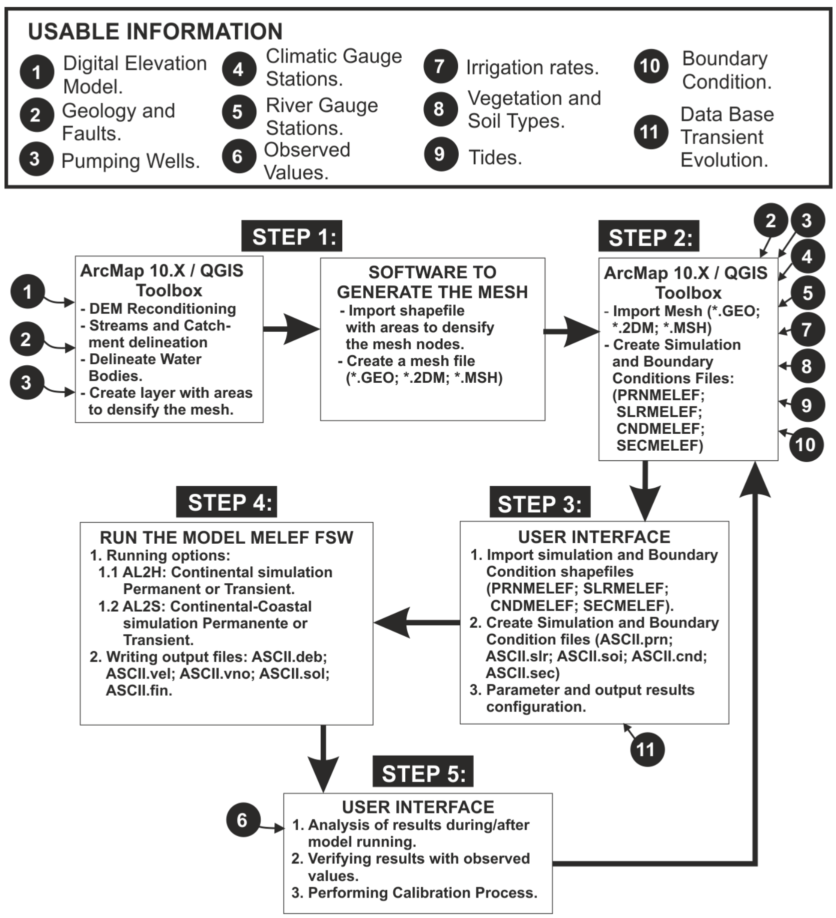


Figure 9: Most relevant steps for a simulation project with MELEF FSW.

For each new project the following steps, shown in Figure 9, are required to prepare the model, run it, calibrate it and verify the simulation:

STEP1: Prepare the hydrological elements of the study area to generate the discrete model (mesh). Before creating the mesh, it is necessary to establish whether the digital elevation model needs to be reconditioned: redefine riverbeds; verify river continuity; update elevation with bathymetry in water bodies; verify location and elevation of dam curtain. In addition, it is necessary to prepare vector layers that delimit the study area and the hydrogeological or topographical features that require a higher density of nodes (lakes, rivers, dams, tunnels, faults, others).

STEP2: Import the mesh to GIS for management of the simulation and boundary conditions. Use the Toolbox MELEF FSW in QGIS or ArcMap to import the mesh (*.geo; *.2dm; *.msh), and generating a point and a polygon layers with the nodes and the elements of the mesh respectively. The mesh in vector layers is required to generate new layers with water uses, rainfall zones, soil and geological properties, surface water conveyance, boundary conditions and gauging sections. These layers with the simulation and boundary conditions are required by the User Interface to generate the simulation files.

STEP3: Generate simulation files to execute the model. Use the User Interface to load the GIS layers with the simulation and boundary conditions to generate the required ASCII files to execute the MELEF FSW model. In addition, use the User Interface to manage all the global properties of the model and prepare the INP file that contains all the instructions to be executed by the model.

STEP4: Run the MELEF FSW numerical model. Use the user interface to generate a simulation folder and prepare here the required files to run the model. During the model execution a CMD window appears to show the percentage of simulation and the model convergence. Use the button *i* to approximate the time required for the simulation to finish. Consider disabling hibernation or automatic restart on your PC, as the simulation may take a couple or several hours solving the discrete model simulation.

STEP 5: Verify results and calibration process. During or after the model run, use the User Interface to generate graphs and maps to analyze the results and calibrate or validate the modeling. To do this, enable the option to compare observed and simulated values in the figures.

2.2 User interface

2.2.1 Create or open a simulation project

When starting the user interface, the **Project Manager** window is displayed (Figure 10), in this window whether you need to create a new simulation project or open an existing one. To open the **Project Manager** go to the **Project > Project Manager** menu.

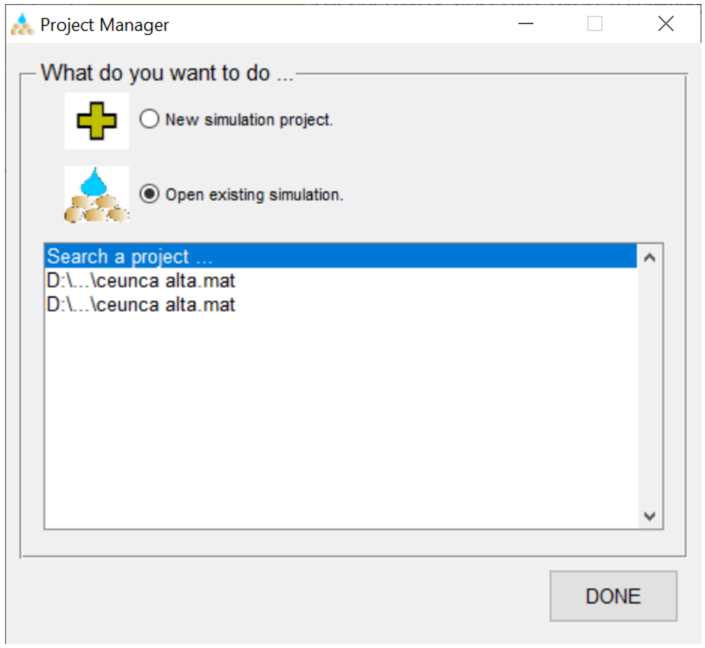


Figure 10: Project Manager window.

Also, you can generate a new simulation project through the menu **Project > New project (Ctrl + N)**, Figure 11.

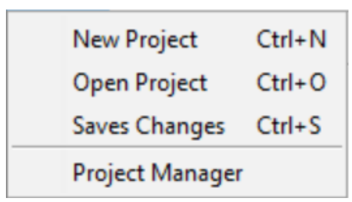


Figure 11: Project Menu: New Project

After generating the project, the interface performs the following actions:

✓Show the name of the project in the lower left part of the main window, Figure 12:

Figure 12: The path and name of the open project is displayed in the main window.

✓ Generates five sub directories in the project folder and a .mat file extension Figure 13. All the project parameter settings are stored in the .mat file. The sub directories allow you to manage all the information from the Geographic Information System, the model simulation results, the database of measured or acquired variables and the results of analysis.

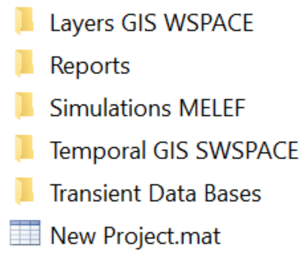


Figure 13: Sub directories generated to manage a simulation project.

2.2.2 Toolbar description

Use the project menu and the shortcut bar to create, open, save or lock the edition of a simulation project.

 Direct access to create a new simulation project.

 Direct access to open a simulation project: select the .mat file to load the project configuration.

 Direct access to save the changes generated in the project's .mat database configuration.

 Direct access to Lock the project edition to work with the user interface tools without modifying the project's .mat database.

 Direct access to open the help manual in HTML version.

2.2.3 Panel 1: Simulation control

Control panel to manage the simulation folders, prepare the simulation files for execution of the model, and obtain information about the time required for the model to finish a simulation. Figure 14 shows the different sections into which the description of this control panel is divided.

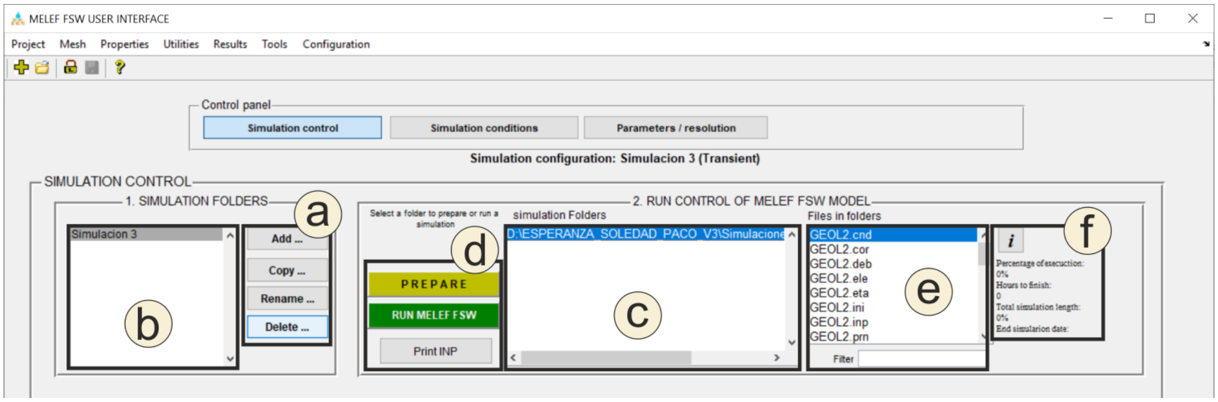


Figure 14: Description of different sections of the Panel 1: Simulation control.

a Buttons to edit the simulation folders:

Add ...: this button creates a new simulation folder in the box **b**.
Copy ...: copies the parameters and variable settings of the simulation folder selected in section **b**.
Rename ...: Renames the simulation folder selected in section **b**.
Delete ...: Deletes the simulation folder selected in section **b**.

b List of simulation folders:

Simulation folders: After create a new simulation folder with the button **Add ...** in section **a** the simulation folders appear in this section. Select a simulation folder to activate panels 2 and 3 of the parameter settings, or execute changes to these with the buttons in section **a**. Additionally, double click on the simulation folder to open it in Windows Explorer.

c List of execution folders paths:

Execution folders paths: After create a new simulation folder with the button **Add ...** in section **a** the execution folders path appear in this section. Select a execution folder path to display the simulation and result files of a simulation in the section **c**. Additionally, double click on the execution folder path to open it in Windows Explorer.

d Buttons to execute the model

Prepare simulation: prepares the execution folder selected in section **c** to run the model. The preparation consists of copying the simulation files and generating the INP file with the configuration of the global parameters of the system resolution. If there are files from a previous run then it updates them (the user is first consulted to confirm this action).

Execute FreshSaltWatershed: Once the execution folder is prepared, the model can be run. Before running the model, the interface verifies that there are no result files from previous simulations with the same name. If there are results files with the same name, the interface consults with the user whether the files should be deleted, renamed, or cancel the model run. In case the execution error file exist, fort8, you will be asked for confirmation to delete it and continue, or cancel the execution. If the fort8 file is not deleted automatically, it will be necessary to delete it manually from the execution folder.

Print INP: Print the INP file directly without having to prepare the simulation or generate the remaining simulation files.

e List of simulation and result files:

Execution and results files: This window displays all simulation and result files contained in the execution folder selected in section **c**. Double click on a file of the list to open it, or select one or more files and press the delete key to delete them.

Filter files: Enter a filter to display the files of interest. You can enter multiple filters separated by a comma or a semicolon, for example the user can use as filter the file extensions "prn, slr, vno, inp" to select all the files with this extensions, or define the file name to select all the files with this name. After write the filter click on the file list section to update it.

f Execution information in progress:

Remaining run time: Select an execution folder path in section **e** and press the ***i*** button to display the run percentage of the simulation, the date the simulation started, and the time remaining to finish.

2.2.4 Panel 2: Simulation conditions

Use this panel to configure the time control, to generate or define simulation files and printing of results. Figure 15 shows the different sections into which the description of this control panel is divided.

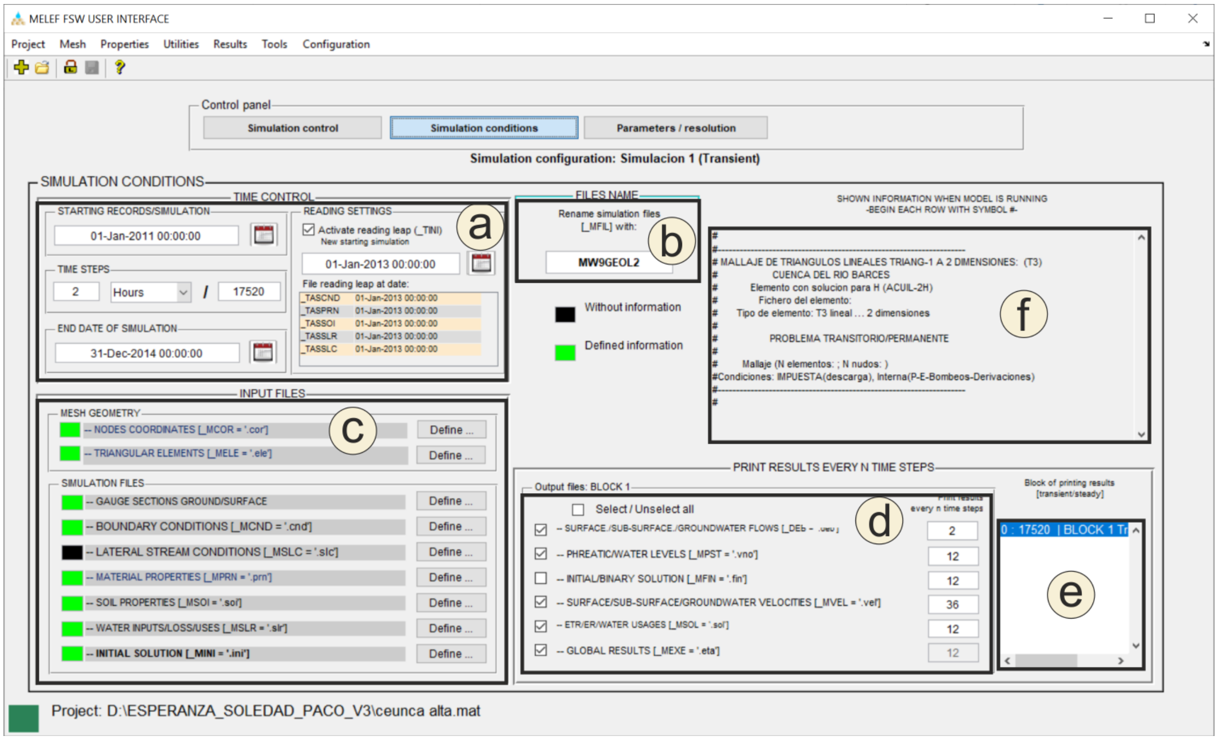


Figure 15: Panel 2: Use this panel to define the start and end simulation time, generate the simulation files and configurate result files.

⊟ a Time control:

Starting records/simulation: Define the start date of simulation records. Write directly in the text box or press the button with the calendar icon to select a date.

Time step: From the drop-down menu select the unit of time, seconds - minutes - hours - days, and then write the modeling time increment to use. In Figure 15 is possible to see that there is a time increment of 2 hours and at the right of the drop-menu it shows the total number of time steps 17520. NOTE: the total number of time steps is automatically evaluated when the start and end dates are defined, and it is evaluated subtracting the initial date from the end date and divided by the time increment.

End date of simulation: Define the end date of the simulation. Write directly in the text box or press the button with the calendar icon to select a date.

Reading settings: Activate reading leap to start the simulation at a different date than the start date of simulation records. The new starting simulation date (_*TINI*) implies a leap in the reading of the _*TASCND*, _*TASPRN*, _*TASSOI*, _*TASSLR*, _*TASSLC* files to the new date to restart the model with the correct simulation and boundary conditions. In addition, by enabling the read jump the INP file resets the total number of time steps to be simulated, although this reduction of time steps will not be visible in the *Time Control*. NOTE: if reading leap is enabled check that the initial solution match the solution of the new starting simulation date (_*TINI*).

b Name of simulation files:

Files name: The interface use this name to rename the simulation files that are copied to the execution folder with the *Prepare* button in the Panel 1: Simulation control. The simulation name cannot exceed 8 characters, and it cannot have blank spaces or special characters such as ",!, &,%, (, /,).

c Input files:

The input files are separated in two blocks: the first block contains the **mesh geometry**; the second block contains the **simulation files**.

Press the **Define...** button, to the right of each input file, to display the dialog box in Figure 16, where you can select one of the three available options: **Select...**; **Generate...**; or **Deactivate...**.

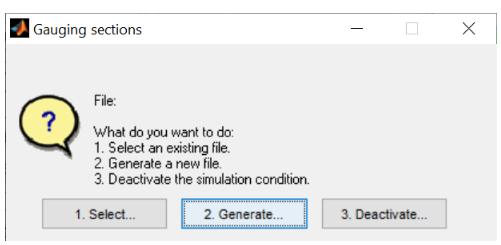


Figure 16: Dialog box to define the simulation files.

1. Select...: this option allows you to select an existing file and reuse it in any simulation.

2. Generate...: this option displays the necessary tool to generate the simulation file.
3. Deactivate...: this option disables the simulation file.

Description of simulation files:

Mesh Geometry: the mesh geometry files can be generated from the .GEO and .2DM files of the *Surface Water Modeling System SMS* program or the format .MSH from *IBER* software. Formats from other programs are also supported (.dat from IWFM Mesh).

- **Nodes coordinates:** file .COR, this file has the XY coordinates of the triangular mesh of elements of three nodes.
- **Triangular elements:** file .ELE, this file hast the nodes numbers of the triangular elements of the mesh.

Simulation files:

- **Gauge sections ground/surface:** file .SEC, this files has the node numbers that conform a gauging section where the model keeps track of the groundwater, subsoil and surface water flows that passes through each section.
- **Boundary conditions:** file .CND, this file has the node numbers with a boundary condition as for example an imposed water level, lateral flux or an open gradient discharge condition.
- **Lateral stream conditions:** file .SLC, this file has the node numbers with a lateral stream condition to define a transient volume of surface water injected or withdrawn.
- **Material properties:** file .PRN, this file has all the mesh nodes with the material properties, the soil topography and the position of the impervious substratum.
- **Soil properties:** file .SOI, this file contains soil-related variables and climate variables that define the model of water uptake from the soil through plant roots. These variables are: Soil Thickness (related with the maximum length of the plant roots); potential values of Evaporation and EvapoTranspiration, Field Capacity, Capillary Fringe of the Soil, Surface Hydraulic Conductivity and Slope.
- **Water inputs/loss/uses:** file .SLR, this file contains the nodes with main water uses, inputs and outputs of the discrete model: Rainfall Zones, Pumping Wells, Water Injections, Surface Water Diversions, Bypass of the Water Diverted, Zones of Irrigation, Dam Floodgates and Spillways.

- **Initial solution:** The initial solution defines the initial fresh water-table level and the soil readily available water in continental simulations, and the initial saltwater location (in meters about sea level) for coastal simulations in the .INI file. The .INI file is required to start transient simulations, see Figure 17.

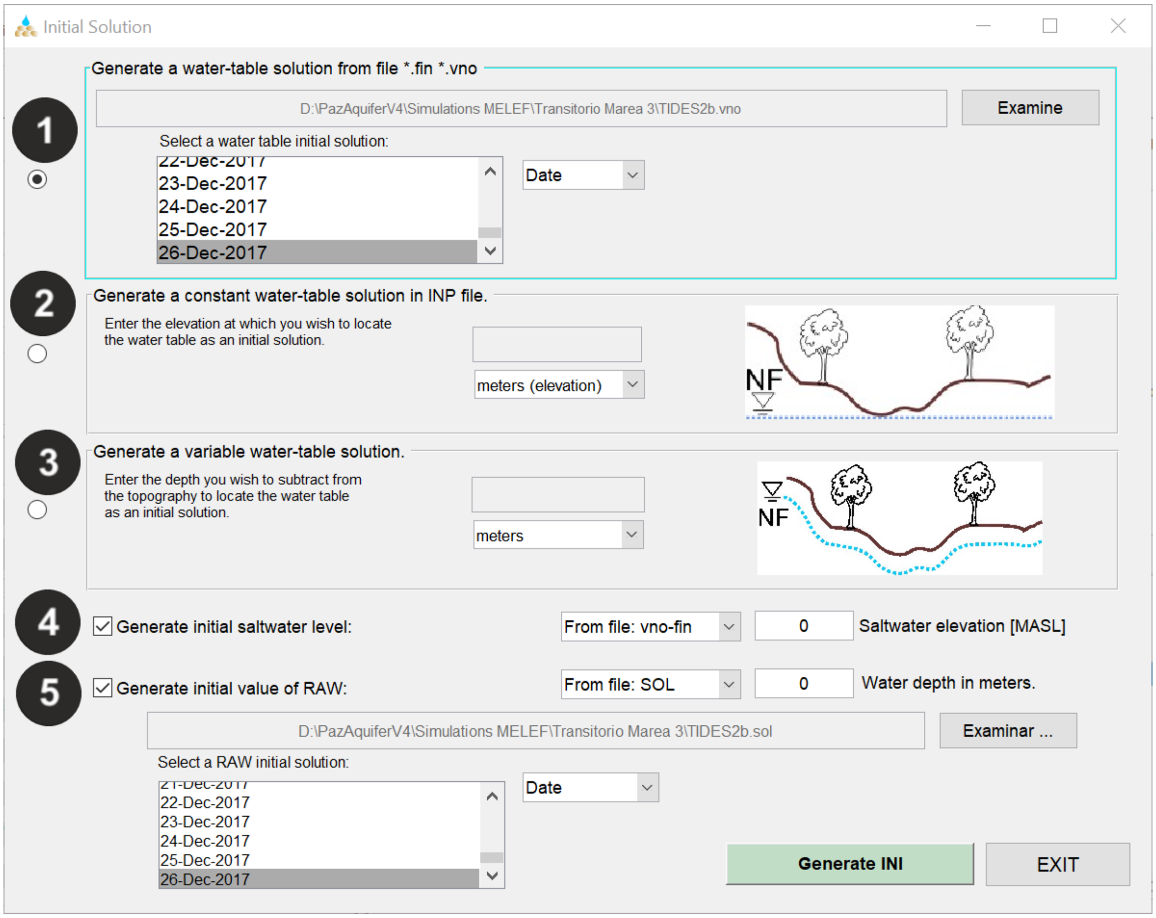


Figure 17: Window for defining the initial solution for the water table, the readily available water in the soil, and the location of salt water. It is possible to define a variable solution from a VNO-FIN-SOL results file, a constant solution, or, in the case of the water table, there is a third option, which consist of using the topography to generate a water table parallel to it.

Options to generate an initial solution

1 Select a file with the extension .FIN or .VNO to generate an initial solution of fresh water levels for all mesh nodes obtained from printed results in these files. Change the time [*Seconds, Days, Date*] in the drop-

down-button. If the *Date* format is selected, a calendar window will appear for the user to select the STARTING RECORDS/SIMULATION date.

2 Enter the water level at which you want to place the initial solution. This solution implies a constant water table level equal to the entire discrete model. This option was designed for use in permanent modeling and does not generate an .INI file, but rather saves the constant value and prints it to the .INP file when the simulation is prepared for execution.

3 Enter a constant depth to the fresh water table (positive value) to define the initial solution parallel to the ground surface (topography). After pressing the *Generate INI* button a PRN file is required to subtract from topography the constant depth.

4 Enable this option only if the simulation uses the *coastal-continental model* (AL2S). This option prints a second column in the .INI file with the initial solution for the elevation of the saltwater position. The initial saltwater solution can be a constant elevation value, or you can choose the option “from file: vno-fin” if you want to use a solution printed in a results file. If there are no result files with the initial solution for saltwater (vno-.fin file), it is recommended to use a constant solution close to mean sea level, or another value close to the level of the saltwater source, so that the model begins to iterate until it finds a solution and prints it in the vno-.fin result file.

5 Enable this option to define an initial solution for the depth (in meters) of Readily Available Water (RAW) in soils. This option works for the *coastal-continental model* (AL2S) and the *continental model* (AL2H). This option prints a second column in the .INI file for AL2H model, or a third column for the AL2S model. The initial solution printed is the soil water content in units of depth in meters. If a .SOL results file exists, you can select it using the "*From file: SOL*" option to select the time step with the RAW value printed in this file as the initial solution, or use the "*Constant RAW value*" option to generate an initial solution with a constant value.

d Configure the print results of the output files:

Printing the results: define the frequency of printing the results, in units of time increments, and check or uncheck the boxes to define which results you want to print. The print frequency of the global ETA log file is related to that of the VNO file, and it is not possible to define different print frequencies, so this option is disabled for these global results.

e Block of global printout configuration:

Block settings: select an item from the box to configure the printing results. Generate the blocks configuration in the Panel 3: Parameters / resolution.

f Model information:

Actual simulation: this information is displayed in the MS-DOS window during the model execution. Edit the information that is displayed by default (optional).

2.2.5 Panel 3: Parameters / resolution

Use this panel to configure the global parameters and resolution parameters of the system. Figure 18 shows the different sections into which the description of this control panel is divided.

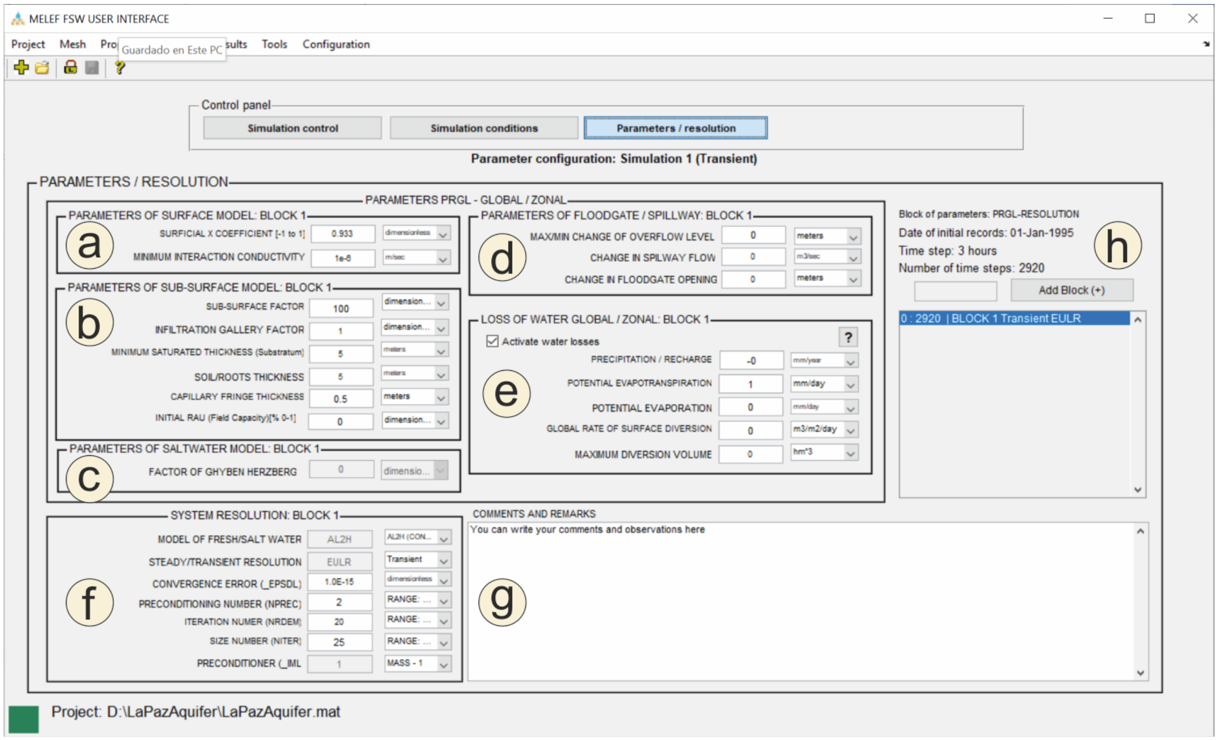


Figure 18: Panel 3: Use this panel to configurate global parameters, activate/deactivate models and modify the system resolution.

a Parameters of surface model:

Surficial X coefficient (-1 to 1): coefficient that define the transmissivity of the surface medium. The values that this coefficient can take is around 0.935, being very sensitive to the variations. Values + (positive) are constant in the whole domain. Values – (negative) would smooth the X value with the ground slope. Therefore, the coefficient X governs the equation for vertically averaged horizontal surface flow of the MELEF FSW model. In the cases of numerical instabilities (or poorly converged solutions) the absolute X value should be slightly reduced, or instead slightly increased if the solutions are solid (highly converged).

Minimum interaction conductivity: The overland water flow should be adapted to the permeable soil surface medium, and it requires a minimum conductivity for this adaptation/interaction (smoothing the Surficial X value with the ground slope) to happen. The value can vary between 1e-1 and 1e-10 (m / sec). This global value also accounts for the maximum hydraulic conductivity of the considered impervious materials (impervious screens, concrete, pavement) below which this interaction would not happen.

b Parameters of subsurface (subsoil) model:

This sub-panel control the subsoil or unsaturated zone of the numerical model, and its global parameters are grouped, although some of them can be defined zonally.

Sub-surface factor: factor that smoothly multiply the groundwater thickness as the water table approaches the soil surface. The values that this factor can take is around 100. Values + (positive) are constant in the whole domain. Values – (negative) would smooth the sub-surface factor with the topography slope. This factor allows the correct transition between groundwater and surface water flows through the subsoil zone.

Factor of infiltration gallery: factor that define if an existing gallery fully infiltrate the water (Factor = 1) or not at all (Factor = 0). Another value between 0 and 1 would adjust the water infiltration capacity of the gallery.

Minimum saturated thickness (substratum): a sinking water table could have a lower position than the impervious substratum, the minimum saturated thickness would allow a minimum (and required) transmissivity to the groundwater flow model.

Soil/Root thickness: global soil thickness as related to the average depth of the plants roots. This thickness is used by the EvapoTranspiration sub-model and the assessment of the sub-surface (subsoil) flow velocities.

Thickness of capillary fringe: the capillary fringe is used to determine the phreatic evaporation from the water table. The
Initial Readily Available Water in soil (RAW) control the water withdrawal capacity of the discontinuous sub-model of EvapoTranspiration.

INITIAL RAW or field capacity (FC) [% 0-FC]: This parameter has two behaviors. The first one is to define a global maximum RAW in soils of the whole modeling area when there is not zonal information (*.SOI file with field capacity and soil thickness) to evaluate the maximum RAW, and the initial RAW value is set to zero by default. The second behavior is when zonal information exists, then this parameter becomes the initial percentage of water in the soil for all zones, and if this value is higher than the zonally percentage of water then the zonally value is set as the initial percentage of water.

c Parameters of Saltwater model:
This parameter defines the general behavior of the coupled continental-coastal model through an immiscible interface approach between fresh and saltwater.

Ghyben-Herzberg factor: when considered (>0) this factor should be between 25 and 35.

d Floodgate/Spillway parameters:
The regulation capacity of floodgates and spillways at the dams of reservoirs requires of three global parameters:

Max/Min Change of overflow level: parameter (in meters) that, on one hand, allows this maximum increase of the floodgates level, and, on other hand, allows this maximum decrease on the topographic level of spillways and weirs or dams of reservoirs. This parameter allows the numerical stability of the regulation facilities (floodgates and spillways) of reservoirs. Usual values should be several meters (1 to 8 meters). A null value (0) would deactivate any defined floodgate or spillway.

Change in spillway flow: Parameter of adjustment that allow to increase or decrease by a constant value [+ m3/sec] the computed flow rate function of the spillway.

Change to floodgate opening: Parameter of adjustment that allow to increase or decrease by a constant value [+ meters] the overflow level of the floodgates. Positive values would generate lower openings and overflows, whereas negative values would generate higher openings and overflows.

e Global/zonal water loss and uses:
Some global parameters describe the main superficial water loss and uses that is characteristics of the zonal file .SLR.

Precipitation/Recharge: define the Global Precipitation or Recharge. The unities could be mm/sec - mm/min - mm/hrs - mm/día - mm/day - mm/month - mm/year. The Interface should transform it in the standard simulation unities m/sec.

Potential EvapoTranspiration: define a Global value of Potential EvapoTranspiration (ETP). In order to activate the ETa-ER sub-model, global or zonal (file .SLR), introduce a value greater than zero.

Potential Evaporation: define the Global value of Potential Evaporation in either of the considered units.

Global Rate of Surface Diversion: define a Global value of surface water Diversion/Detention rate coefficient (m3/m2/day). A zero value could allow Zonal Diversion/Detention

Max. Diversion Volume: this volume (>0 hm3) activates the Diversion/Detention sub-model, Global or Zonal, but it would deactivate when this volume is reached.

IMPORTANT NOTES:
the following working scheme would apply:
1) The global Precipitation would be added to the zonal water recharge of file .SLR;
2) The ETa-ER sub-model activates whether this Global ETP value is higher than zero;
3) The Zonal ETP and EP values (from the SOI file) prevail over these Global Potential values;
4) The Global Rate of Surface Diversion/Detention (>0 m3/m2/day) prevail over the corresponding Zonal values, which in this case would be interpreted as Zonal withdrawal (m3/m2/sec);
5) The Global Maximum Diversion/Detention Volume (>0) activates the Diversion/Detention sub-model.

System Resolution:

Edit the settings for the type of model to be solved, continental (freshwater) or coupled continental/coastal (freshwater/saltwater), the stationary or transient water regime, and the parameters of the GMRS solver algorithm.

Fresh/Saltwater Model: select the continental (AL2H) or the coupled continental/coastal (AL2S) model.

Steady/Transient Resolution: select a Steady or a Transient resolution.

Convergence Error (_EPSDL): indicate the Error of the sought convergence, default value should be 1.0E15. During the system resolution, the found convergence error should keep smaller than 1.

Preconditioning number (NPREC): parameter of the GMRS resolution algorithm that indicates the number of evaluations in a same time step (suggested value to use: 3)

Iteration number (NRDEM): parameter of the GMRS resolution algorithm, that indicates the number of iterations in order to reach the desire convergence (for general problems this number do not need to be much higher than 25, suggested value to use: 14).

Size number (NITER): parameter of the GMRS resolution algorithm, that indicates the size of the sub-matrix used to resolve the general numerical system (for general problems this number do not need to be much higher than 20, suggested value to use: 14).

Preconditioner (_IML): select 0 (unity) or 1 (Masse), for a faster convergence better use Preconditioner 0.

g Parameters Block PRGL-RESOLUTION:
Add a new block of configuration of Global parameters, System resolution (Panel 2) and/or Printout results (Panel 3).

Add block (+): add a new block of configuration introducing, if desired, the new time step at the left of the button. Select the block from the list below to configure, if desired, the new Global parameters, the System resolution and/or the Printout results. Create new blocks of configuration to change global parameters with time.

h Add comments and remarks:
Comments and remarks: add any comment or description that should be useful to characterize the actual simulation (optional).

2.2.6 Menus and Tools

Introduction to the configuration and results analysis tools available in the menu bar of the User Interface. Figure 19 shows the different sections into which the description of sub-menus of the menu toolbar.

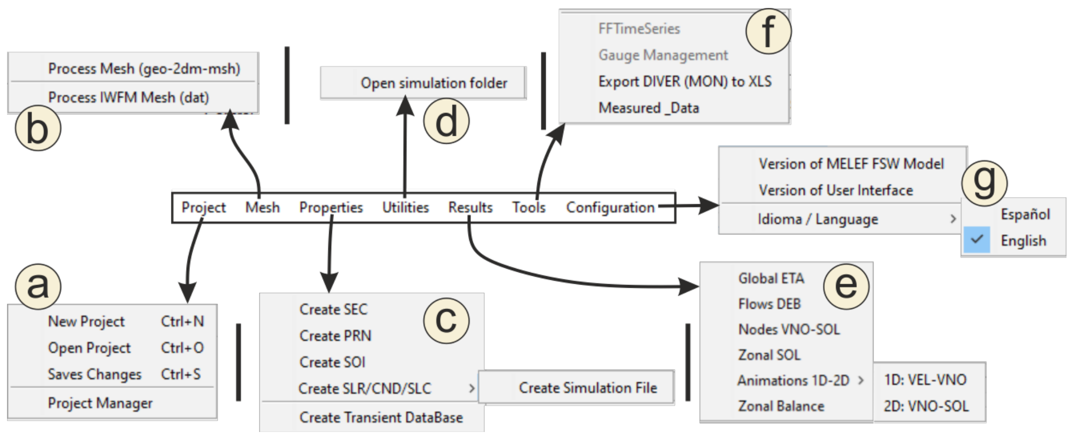


Figure 19: Description of the configuration and results analysis tools contained in the menu toolbar of the MELEF Interface.

Usage of the tool sections marked with alphabetic labels in Figure 19:

2.2.6.1 Menu Project

In this menu you will find the tools to create, open or save a project, as well as, a project manager with the latest open projects.

New Project Ctrl+N: Opens a window to define the path and name of the new project to be generated. Use the keyboard shortcut Ctrl+N for a direct access.

Open Project Ctrl+O: Opens a window to search a project and open it. Use the keyboard shortcut Ctrl+O for a direct access.

Saves Changes Ctrl S: Saves the changes made to the actual project. Use the keyboard shortcut Ctrl+S for a direct access.

Project Manager: Opens the project manager window to select from a list the recently opened projects to open it.

2.2.6.2 Menu Mesh

b

In this you will find some tools to import the finite element mesh from different software's and different mesh formats.

2.2.6.2.1 Process Mesh (geo-2dm-msh)

Use this tool to process the files .GEO or .2DM, of the *Surface Water Modeling System SMS program,* or the format .MSH from *IBER* software to generate the simulation files with the mesh geometry .COR and .ELE. Figure 20 shows the different sections into which the description of this tool.

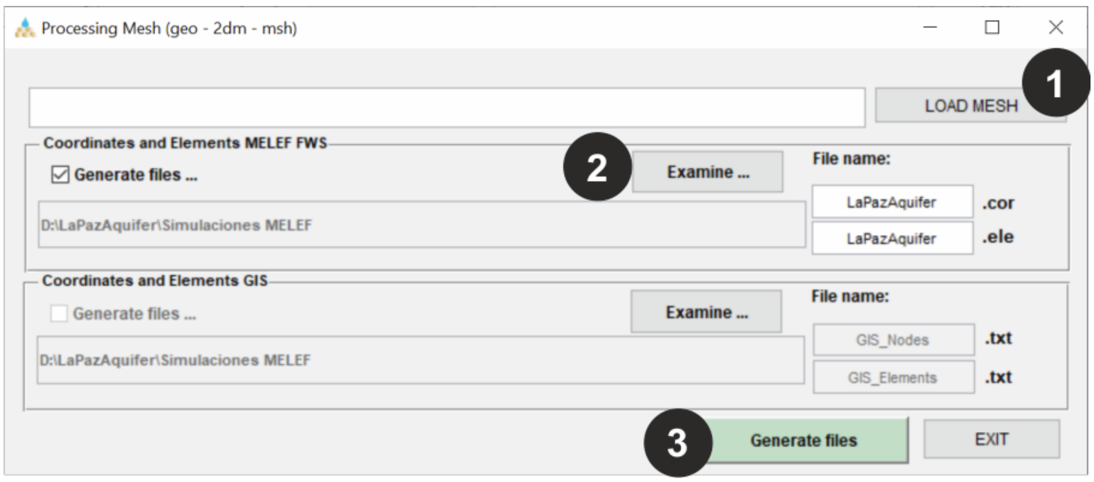


Figure 20: Tool to import the mesh and generate the simulation files .COR and .ELE.

Usage of the tool sections marked with numeric labels in Figure 20:

1 **LOAD MESH:** Display the window explorer to select the .GEO - .2DM - .MSH format file. The tool verifies that the mesh is made of linear triangular elements (three nodes per element), although it can process a mesh of quadratic triangular elements of six nodes and generate the geometry in text files for GIS. NOTE: the option to generate GIS files will be deprecated in future versions because the GIS Toolbox already include a tool to import the mesh and create GIS layers directly.

2 **Coordinates and Elements MELEF FSW:** Check the box *Generate files ...* to generate the files .COR and .ELE with the names defined in *File name*

section. Click the button **Examine ...** to select the path where the files should be generated. In the same manner use the section to create the *Coordinate and Elements GIS*. NOTE: the output path to the folder of simulation and the name file are filled automatically if a simulation project is open.

3 **Generate files:** Click the button to create the files and store the file path in the .mat database of the interface.

2.2.6.2.2 Process IWFM Mesh (dat)

use the explanation of the Figure 20 to use this tool.

2.2.6.3 Menu Properties

C

In this menu you will find different tools to generate simulation files from GIS layers. These GIS layers must be previously generated with the GIS Toolbox.

2.2.6.3.1 Create SEC

Use this tool to import the database of the shapefile SECMELEF with the nodes that conform the gauging sections.

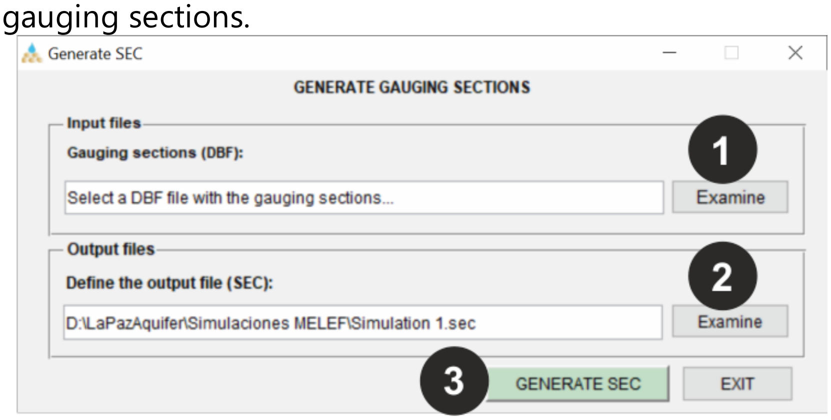


Figure 21: Tool to import the DBF of a the shapefile SECMELEF created with Toolbox GIS and create simulation file with the gauging sections.

Usage of the tool sections marked with numeric labels in Figure 21:

1 **Examine:** Display the windows explorer to select the file SECMELEF.dbf (or similar name) and create the simulation file .SEC with the gauging sections.

2 **Examine:** Change the output path and name file if you require to change the automatic filled data.

3 **GENERATE SEC:** Click here to execute the tool and generate the simulation file SEC with the gauging sections.

2.2.6.3.2 Create PRN

Use this tool to generates the nodal properties related with Geology and Topography. The source of information is the database file (.DBF) associated to the shapefile PRNEMELEF (or similar name file) created with the GIS Toolbox. If any property change over time, then use a transient database to define their temporal evolution. Finally this tool creates the simulation file PRN.

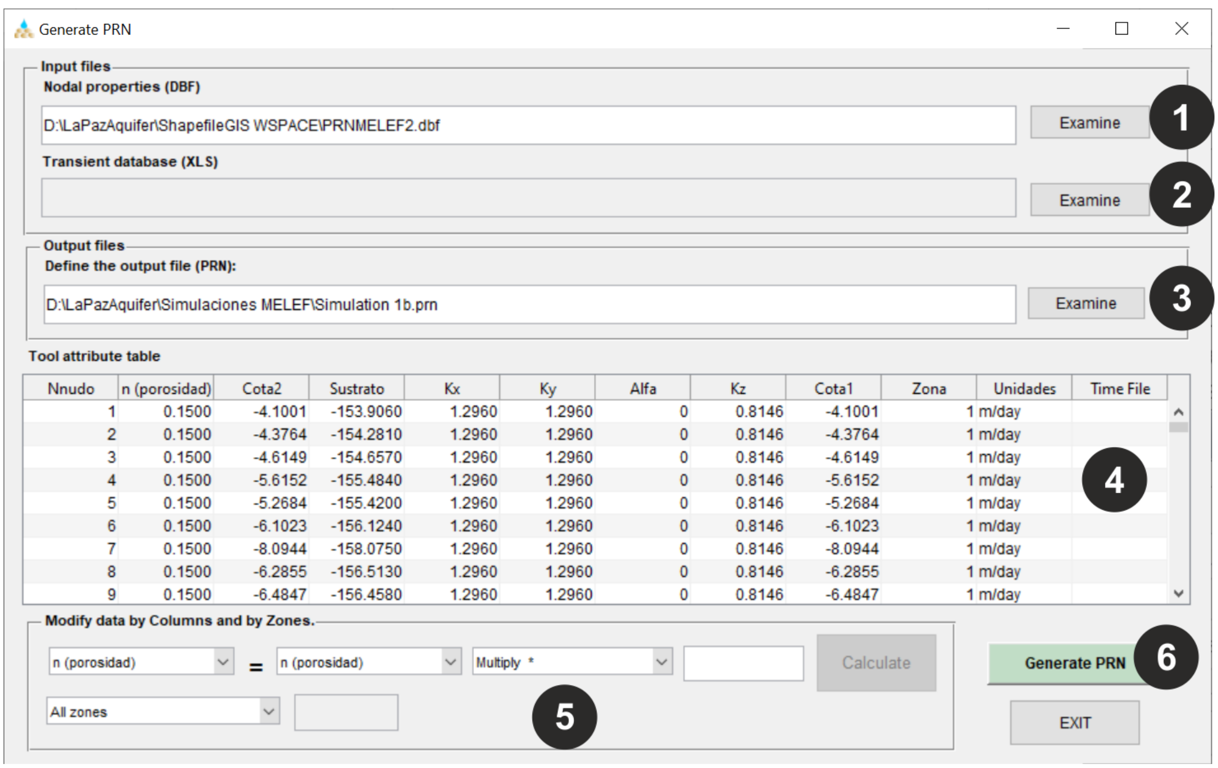


Figure 22: Tool to generate the geological and topographic nodal properties of the discrete model. The tool implements the attribute table DBF file of the shapefile PRNMELEF generated with the GIS Toolbox.

Usage of the tool sections marked with numeric labels in Figure 22:

1 **Nodal Properties (Examine):** Click the button to display a window explorer and select the file .DBF (attribute table) associated with a shapefile PRNMELEF (or similar name). After select the DBF file, the first step executed by the tool is verifying the structure of the attribute table to search for the predefined column names; once the attribute table is verified, all the attribute table is loaded in the attribute table of the tool (see section with label 4).

2 **Transient Database (Examine):** Click the button to display a window explorer and select the file .XLS - .XLSX (attribute table) with the temporal evolution of parameters related with the Geology or Topography. If the column *Time File* in the DBF attribute table is empty, then automatically this section is disabled because a transient behavior is not required. NOTE: use the tool *Create Transient Database* to generate the Excel file structure automatically with the attribute table DBF file.

3 **Define the output PRN file (Examine):** Click the button to display a window explorer to define the path and name file of the simulation file PRN. The tool automatically fills this information using the folder simulation path and name to define the output path and name file.

4 **Tool attribute table:** If the attribute table of the shapefile PRNMELEF is valid, then the information is loaded to the tool attribute table. The variables in the attribute table are the following:

1. NNUDO: Node identification number.
2. N: Effective porosity of materials that can be in units of percentage [%] or [0-1].
3. COTA1: Defines the elevation of topography of each node.
4. COTA2: Defines the elevation of topography of each node. Change this value to activate a secondary topography and enable the simulation of tunnels or caverns with a known base elevation. Consider it is only an approximation in two dimensions where the ceiling of the tunnel or cavern is directly the soil layer.
5. SUSTRATO: Elevation of the impervious substratum that defines the lower limit of the simulated aquifer. Below the substratum the groundwater flows with a minimum constant transmissivity.

6. KX: Hydraulic conductivity of materials in the X axis in units of m/sec if it is not defined in the field UNIDADES.
7. KY: Hydraulic conductivity of materials in the Y axis in units of m/sec if it is not defined in the field UNIDADES.
8. KZ: Hydraulic conductivity of the soil in the Z axis, It could be considered as the infiltration capacity, in units of m/sec if it is not defined in the field UNIDADES.
9. ALFA: Angle of anisotropy of the hydraulic conductivity in degrees. It controls the direction of X and Y axes measured from north and counterclockwise.
10. ZONA: This field identifies the nodes that pertain to the same zone.
11. UNIDADES: Units of hydraulic conductivity parameters, keep this field empty to use the default units of meters per second (m/s) or define other units like meters per day (m/d).
12. TIME_FILE: This field has the labels required to link a column of an Excel file with the nodes and create a transient behavior of the nodal properties. Create the Excel file previously with the tool *Create Transient Database*..

5 **Modify the tool attribute table:** Use this section to modify the property values by column or by zone. First, from left to right, in the pop-up drop-down menus, select the column you want to modify, the base column to carry out the operation, the type of operation and its magnitude and then click on the *Calculate* button. The changes in the tool attribute table are saved once the button *Generate PRN* is pressed only if there is a project open.

6 **Generate PRN:** Press the button to write the simulation file PRN in the folder of simulations (default path), and if there is a project open the tool saves the change made in the tool attribute table and save the PRN file path in the MAT database of the project.

2.2.6.3.3 Create SOI

Use this tool to generate nodal properties related with the Soil. The source of information is the database file (.DBF) associated to the shapefile SOIEMELEF (or similar name file) created with the GIS Toolbox. If any property change over time, then use a transient database to define their temporal evolution. Finally this tool creates the simulation file SOI.

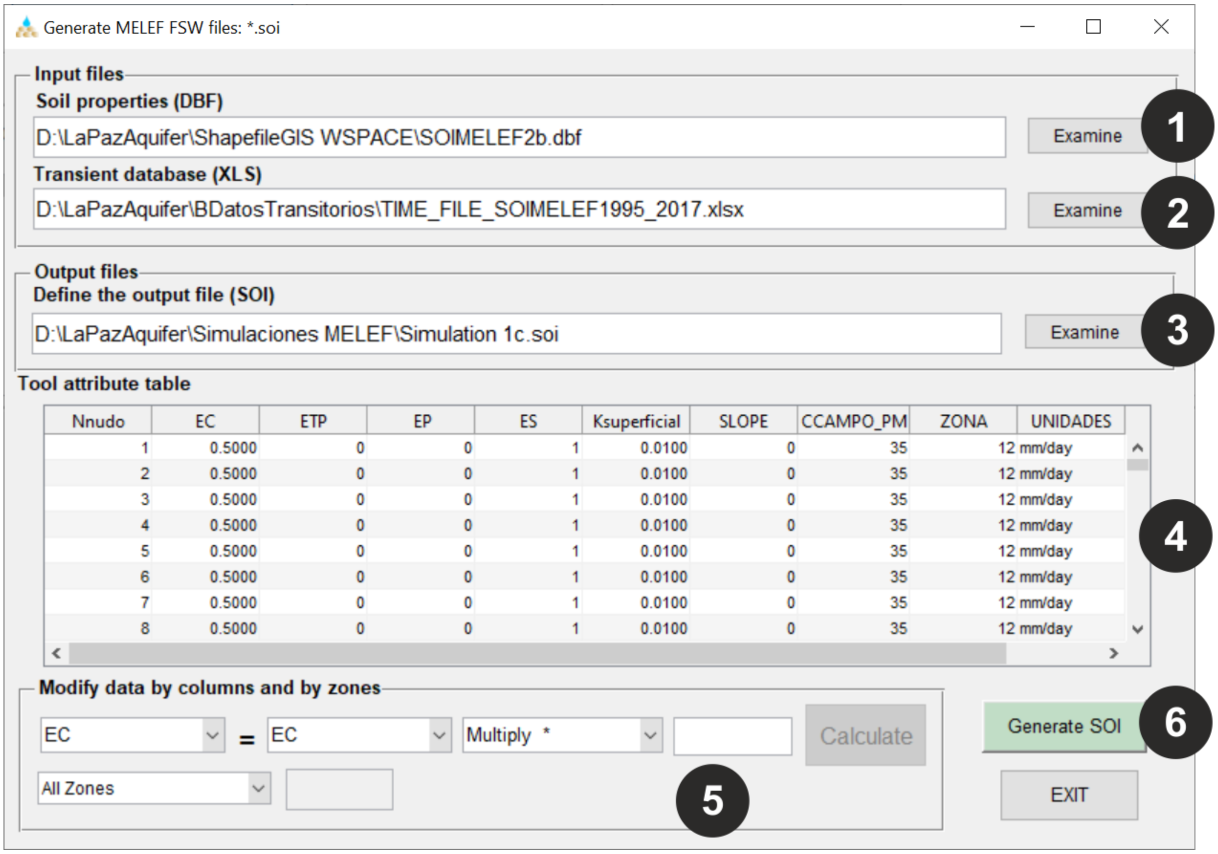


Figure 23: Tool to generate the soil nodal properties of the discrete model. The tool implements the attribute table DBF file of the shapefile SOIMELEF generated with the GIS Toolbox.

Usage of the tool sections marked with numeric labels in Figure 23:

1 **Soil Properties (Examine):** Click the button to display a window explorer and select the file .DBF (attribute table) associated with a shapefile SOIMELEF (or similar name). After select the DBF file, the first step executed by the tool is verifying the structure of the attribute table to search for the predefined column names; once the attribute table is verified, all the attribute table is loaded in the attribute table of the tool (see section with label 4).

2 **Transient Database (Examine):** Click the button to display a window explorer and select the file .XLS - .XLSX (attribute table) with the temporal evolution of parameters related with the Soil and climate (usually EP and ETP). If the column *Time File* in the DBF attribute table is empty, then automatically this section is disabled because a transient behavior is not required. NOTE: use

the tool Create Transient Database to generate the Excel file structure automatically with the attribute table DBF file.

3 **Define the output SOI file (Examine):** Click the button to display a window explorer to define the path and name file of the simulation file SOI. The tool automatically fills this information using the folder simulation path and name to define the output path and name file.

4 **Tool attribute table:** If the attribute table of the shapefile SOIMELEF is valid, then the information is loaded to the tool attribute table. The variables in the attribute table are the following:

1. NNUDO: Node identification number.
2. EC: Capillary Fringe Thickness [Meters].
3. ETP: Potential EvapoTranspiration. The units can be defined by user in the field [UNIDADES] (MM/DAY; MM/DIA, check manual for more units or use default units m/sec if field UNIDADES is empty.
4. EP: Potential Evaporation from surface water or bare soils, see ETP for units.
5. ES: Soil Thickness related with the maximum depth of vegetation roots [Meters].
6. KSUPERFICIAL: This value defines the hydraulic conveyance of water on water bodies, streams or overland flow areas. The values varies between 1e-2 to 1e-7, where higher values defines more hydraulic conveyance (water bodies and rivers: 1e-2 to 1e-5; hillslopes with different degrees of vegetation: 1e-6 to 1e-8).
7. SLOPE: Slope in degrees of mesh nodes.
8. CCAMPO_PM: Percentage of Readily Available Water in soils that is available for EvapoTranspiration of vegetation [%].
9. ZONA: This field identifies the nodes that pertain to the same zone.
10. UNIDADES: Units of hydraulic conductivity parameters, keep this field empty to use the default units of meters per second (m/s) or define other units like meters per day (m/d).
11. TIME_FILE: This field has the labels required to link a column of an Excel file with the nodes and create a transient behavior of the nodal properties. Create the Excel file previously with the tool *Create Transient Database*.

5 **Modify data by columns and by zones:** use this section to modify the property values by column or by zone. First, from left to right, in the pop-up drop-down menus, select the column you want to modify, the base column to carry out the operation, the type of operation and its magnitude and then click on the *Calculate* button. The changes in the tool attribute table are saved once the button *Generate SOI* is pressed only if there is a project open.

6 **Generate SOI:** Press the button to write the simulation file SOI in the folder of simulations (default path), and if there is a project open the tool saves the change made in the tool attribute table and save the SOI file path in the MAT database of the project.

2.2.6.3.4 Create SLR/CND/SLC

Use this tool to generate simulation files of water uses and rainfall (SLR), boundary conditions (CND) and stream lateral condition (SLC). The source of information is the database file (.DBF) associated to the shapefile SLRMELEF/CNDMELEF/SLCEMELEF (or similar name file) created with the GIS Toolbox. If any property change over time, then a transient database is required to define their temporal evolution. Finally this tool creates the simulation files SLR/CND/SLC.

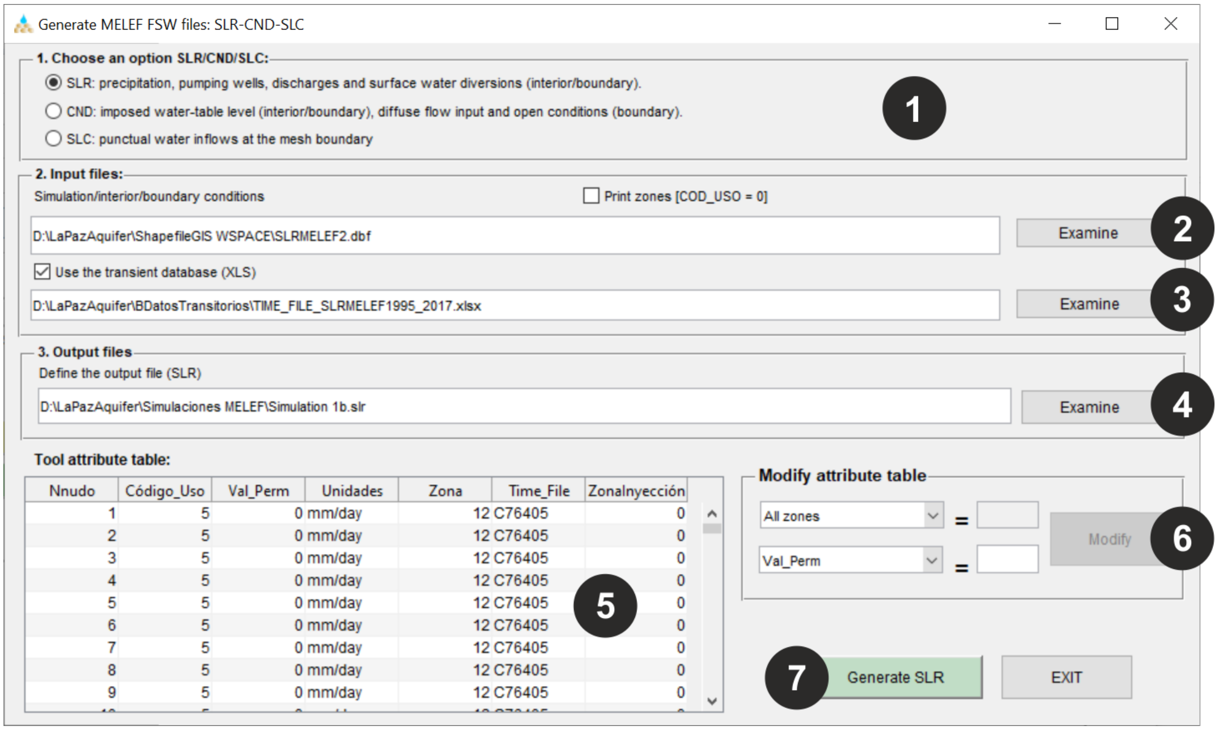


Figure 24: Tool to generate the simulations files SLR, CND or SLC. The tool implements the attribute table DBF file of the shapefile SLRMEEF, CNDMELEF, SLCMELEF generated with the GIS Toolbox.

Usage of the tool sections marked with numeric labels in Figure 24:

1 **Select one option SLR/CND/SLC:** select one of the SLR/CND/SLC options to generate the corresponding file. The tool loads the file paths, and the attribute table values from the simulation project database.

2 **Select an attribute table file DBF of a shapefile (Examine):** click on the button to display a window explorer and select an attribute table file DBF of the corresponding shapefile SLRMELEF/CNDMELEF/SLCMELEF (or similar name). After select the DBF file, the first step executed by the tool is verifying the structure of the attribute table to search for the predefined column names; once the attribute table is verified, all the attribute table is loaded in the attribute table of the tool (see section with label 5).

3 **Database XLS (Examine):** click the button to display a window explorer and select the file .XLS - .XLSX (attribute table) with the temporal evolution of

parameters related with the water uses, boundary condition or stream lateral condition. If the column *Time File* in the DBF attribute table is empty, then automatically this section is disabled because a transient behavior is not required. Take into account that, the extraction and water injection flows (code 4 in GIS for pumping and injection) is divided by the total number of nodes with the corresponding action area, in such a way that the .SLR file prints the flow divided by the number of involved nodes. NOTE: use the tool *Create Transient Database* to generate the Excel file structure automatically with the attribute table DBF file.

4 **Define the Output file SLR/CND/SLC (Examine):** Click the button to display a window explorer to define the path and name file of the simulation file SLR/CND/SLC. The tool automatically fills this information using the folder simulation path and name to define the output path and name file.

5 **Tool attribute table:** if the attribute table of the shapefile SLRMELEF/CNDMELEF/SLCMELEF is valid, then the information is loaded to the tool attribute table. The variables in the attribute table are the following:

1. NNUDO: Node identification number.
2. CODIGO_USO: this code defines what the node is simulating: rainfall, pumping well, water injection, surface water diversion, an imposed fresh/salt water level as boundary condition, imposed surface flow as lateral stream condition.
3. VAL_PERM: Permanent value of the fresh water use, boundary condition or stream lateral condition.
4. VAL_PERM2: Permanente value of saltwater for the codes of use [11, 21 ,41].
5. UNIDADES: Units of VAL_PERM (permanent value) field, if this field is empty then the default units are considered m/sec for SLR; m for CND and m3/m2/day for SLC.
6. ZONA: This field identifies the nodes that pertain to the same zone.
7. TIME_FILE: This field has the labels required to link a column of an Excel file with the nodes and create a transient behavior of the nodal properties. Create the Excel file previously with the tool *Create Transient Database*.
8. ZONA_INYECCION: this field identifies the zone to inject water generated by other zones with a water use condition or dam management.

6 **Modify attribute table:** Use this section to modify the *Val_Perm* field by zones with a permanent value. Changes to the table are stored once the corresponding SLR/CND/SLC file is generated.

7 **Generate SLR/CND/SLC:** press the button to generate the simulation file SLR/CND/SLC and save changes made in the tool attribute table.

2.2.6.3.5 Transient Database

Use this tool to generate the structure of the transient Excel database from the attribute table DBF file related with the the simulation shapefiles: SLRMELEF/CNDMELEF/SLCMELEF, PRNMELEF and SOIMELEF.

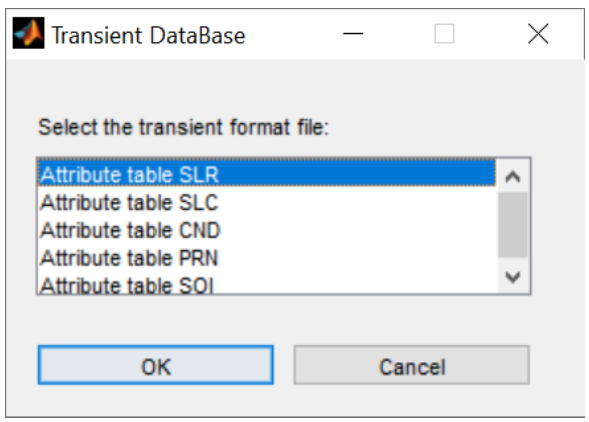


Figure 25: Tool to create the transient database structure in an Excel file using the attribute table DBF file associated with the shapefile generated with the GIS toolbox.

Usage of the tool sections in Figure 25:

1. First select the transient database format you require to generate, the options are the attribute table DBF files of SLRMELEF; CNDMELEF; SLCMELEF; PRNMELEF; and SOIMELEF.
2. Press the *OK* button and define the output path and name of the Excel file.
3. Open the Excel file (the interface tries to open it automatically) and fill in the information with the transient evolution of the parameters: NOTE: if some parameter does not need a transient behavior then delete the parameter column or define a constant value for it. In case some parameter is removed from the Excel database, then the interface will use the values of the columns VAL_PERM from the tool attribute table.

4. Use the Excel file generated as transient database in the corresponding tool to generate the simulation files.

2.2.6.4 Menu Utilities

d

Use this menu as a quick access to the project folders.

2.2.6.4.1 Open simulation folder

Use this sub-menu as quick access to the folder of simulations of the project in a window explorer. There are other ways to access the simulation folder, which are through *Panel 1: Simulation control* by double clicking on the simulation folder list.

2.2.6.5 Menu Results

e

In this menu you will find different tools to analyze all the results generated by the MELEF FSW numerical model in a global and zonal manner. The Global Analysis (Global ETA Tool) generates graphs of results with the evolution of inflows and outflows, which includes results of climate, water uses and fresh/saline water reserves, among other descriptive variables of the study area. On the other hand, the zonal analysis allows comparing the results of any part of the discrete model with observed values and analyze the evolution of flows (Flows DEB), water levels (Nodes VNO-SOL), evapotranspiration rates (Nodes VNO-SOL; 2D: VEL-VNO-SOL) and velocities (2D: VEL-VNO-SOL), all of these in the domain of surface, sub-surface and groundwater. As well as, analysis of the position of fresh-saltwater interface if there are results of the continental-coastal interaction model. In addition, this tools permit to generate 2D maps with the configuration of groundwater levels and surface water depths, profiles with topography and the water level evolution, and create videos with these results.

2.2.6.5.1 Global ETA

Use this tool to analyze the global results of the climatic and hydrological variables in 1D graphics, as well as export reports with all the information in an Excel format (XLS).

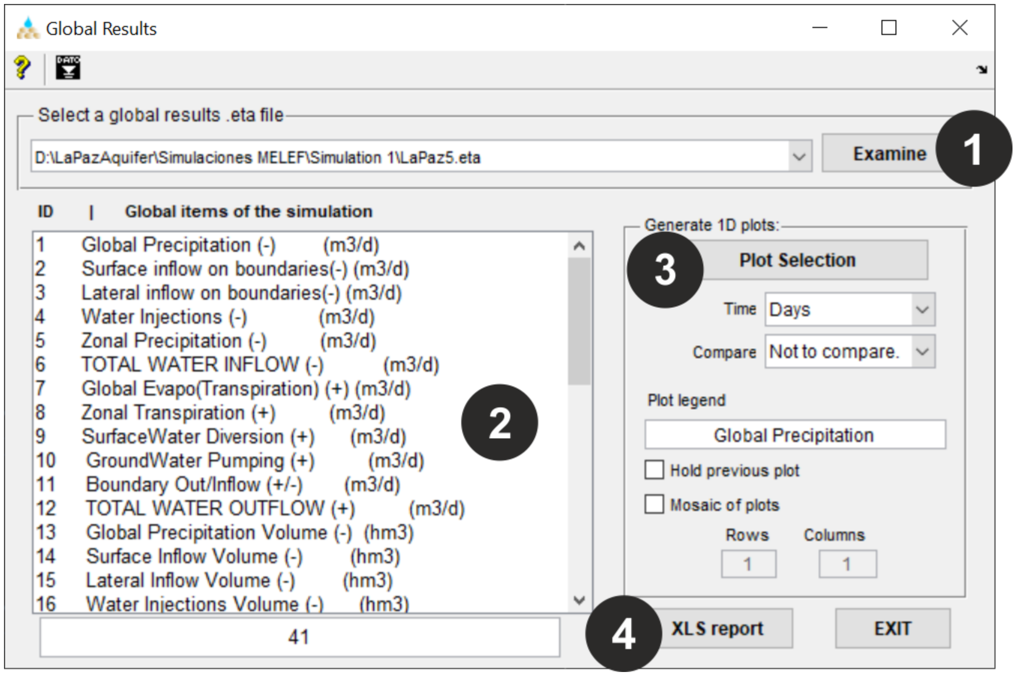


Figure 26: Tool to analyze the global results in the ETA output file of the model.

Usage of the tool sections marked with numeric labels in Figure 26:

1 **Select a global results ETA file** (**Examine)**: Click the button to display a window explorer and select the file .ETA with the global results of the simulation. Automatically the tool opens the folder of simulations and the user needs to open the execution folder and select the ETA file to analyze.

2 **Global items of the simulation:** Once the ETA file is indexed the list of global items is created in this section. Click over one or more global items before plot them. The ETA file hast two versions of results, the first with 41 variables when only continental freshwater flow is simulated (AL2H model), or 54 variables when both continental and coastal models are simulated (AL2S) where both freshwater and saltwater are included as described in the following table:

AL2H	AL2S	Description
1	1	**Global Precipitation (-) (m3/day):** Rainfall intensity imposed globally in the INP file (User Interface).
2	2	**Surface inflow on boundaries (-) (m3/day):** Lateral contribution of rivers or surface water transfer imposed in the CND file (GIS Toolbox).

3	3	**Lateral inflow on boundaries (-) (m3/day):** Lateral diffusive contribution of groundwater imposed in the CND file (GIS Toolbox).
4	4	**Water Injections (-) (m3/day):** Total water flows injected (water spills) imposed in the SLR file (GIS Toolbox).
5	5	**Zonal Precipitation (-) (m3/day):** Precipitation intensity imposed zonally in the SLR file (GIS Toolbox).
6	6	**Zonal Recharge (-) (m3/day):** Vertical water recharge to the aquifer (internally evaluated).
7	7	**TOTAL WATER INFLOW (-) (m3/day):** Total summary of contributions (items 1 to 5) from rivers, lateral diffusive, zonal surface water and rainfall (internally evaluated).
8	8	**Global Evapo(Transpiration) (+) (m3/day):** Global evaporation intensity (+transpiration in transition zones of surface/ground water) (internally evaluated). Potential value imposed globally in the INP file (User Interface).
9	9	**Zonal Transpiration (+) (m3/day):** Zonally evaluated transpiration intensity (internally evaluated). Potential value imposed zonally in the SOI file (GIS Toolbox).
10	10	**SurfaceWater Diversion (+) (m3/day):** Evaluated surface water diversion flow rate (global and/or zonal) (internally evaluated). Potential value imposed globally in the INP file and zonally in the SLR file (User Interface and GIS Toolbox).
11	11	**GroundWater Pumping (+) (m3/day):** Groundwater withdrawal flows imposed zonally in the SLR file (GIS Toolbox).
12	12	**Boundary Out/Inflow +/- (m3/day):** Summary of exits/entries at open boundaries (internally evaluated).
13	13	**TOTAL WATER OUTFLOW (+) (m3/day):** Summary of total exits of items 8 to 12 (internally evaluated).
14	14	**Global Precipitation Volume (-) (hm3):** Global Volume of rainfall (cumulative value).
15	15	**Surface Inflow Volume (-) (hm3):** Volume of lateral contribution to rivers or surface water (cumulative value).
16	16	**Lateral Inflow Volume (-) (hm3):** Volume of lateral diffusive contribution to groundwater (cumulative value).
17	17	**Water Injections Volume (-) (hm3):** Volume of water injections to the aquifer (cumulative value).
18	18	**Zonal Precipitation Volume (-) (hm3):** Volume of surface zonal precipitation (cumulative value).

19	19	**Zonal Recharge Volume (-) (hm3):** Volume of vertical recharge to the aquifer (cumulative value).
20	20	**TOTAL INFLOW VOLUME (-) (hm3):** Total volume of entries/inflows (cumulative value).
21	21	**Global Evapo(Transpiration) Volume (+) (hm3):** Volume of global evaporation (+transpiration in transition zones of surface/ground water) evaluated from surface water bodies (accumulated value).
22	22	**Zonal Transpiration Volume (+) (hm3):** Zonally evaluated vegetation transpiration volume (cumulative value).
23	23	**Diverted Water Volume (+) (hm3):** Volume of surface water diverted (cumulative value).
24	24	**Pumped Water Volume (+) (hm3):** Volume of groundwater pumping extraction (cumulative value).
25	25	**Boundary In/Outflow Volume +/- (hm3):** Volume of surface/ground water exits/entries at open boundaries (cumulative value).
26	26	**TOTAL OUTFLOW VOLUME (+/-) (hm3):** Balance of all the exits/outflows volumes (cumulative value).
27	27	**TOTAL IN/OUTFLOW VOLUME (-/+) (hm3):** Balance of all entries/exits volume (item 20+ item 26) of water.
28	28	**Continental Basin Area (km2):** Continental surface of the discrete model.
29	29	**SurfaceWater Area (km2):** Evaluated area of surface water bodies.
30	30	**Coastal Basin Area (km2):** Evaluated area of coastal seawater.
31	31	**OverlandFlow Area (km2):** Evaluated area of existing overland flow.
32	32	**OverlandFlow thickness (m):** Thickness of overland water zonally evaluated in m^3/m^2 of soil surface.
33	33	**Global Precipitation thickness (-) (m):** Precipitation thickness imposed globally in the INP file.
34	34	**Zonal Precipitation thickness (-) (m):** Precipitation thickness imposed zonally in the SLR file.
35	35	**Zonal Recharge thickness (-) (m):** Zonally evaluated vertical recharge thickness to the aquifer.
36	36	**Evapo(Transpiration) thickness (+) (m):** Evaporation thickness evaluated zonally from surface water bodies (+transpiration in transition zones of surface/ground water)

37 37 **Zonal Transpiration thickness (+) (m):** Zonally evaluated vegetation transpiration thickness.

38 38 **TOTAL IN/OUTFLOW THICKNESS (-/+) (m):** Balance of all input/output of water thickness (sum of items 33, 34, 36, 37).

39 **Coastal FreshSurfaceWater Volume (hm3):** Volume of fresh surface water evaluated in the coastal area.

40 **Coastal SaltSurfaceWater Volume (hm3):** Volume of salt surface water evaluated in the coastal zone.

41 **Coastal FreshGroundwater Volume (hm3):** Volume of fresh groundwater evaluated in the coastal zone.

42 **Coastal SaltGroundwater Volume (hm3):** Volume of salt groundwater evaluated in the coastal zone.

43 **COASTAL FRESH-WATER VOLUME (hm3):** This variable sums the fresh surface-water and fresh groundwater volumes evaluated in the coastal zone.

44 **TOTAL COASTAL WATER VOLUMES (hm3):** This variable sums the volume of freshwater and saltwater, both surface-water and groundwater, evaluated in the coastal zone.

39 45 **Continental SurfaceWater Reserve (hm3):** Volume of fresh surface-water evaluated in the continental zone.

40 46 **Continental Groundwater Reserve (hm3):** Volume of fresh groundwater evaluated in the continental zone.

41 47 **TOTAL CONTINENTAL WATER RESERVE (hm3):** This variable sums the volume of freshwater, both surface-water and groundwater, evaluated in the continental zone.

48 **FRESH SURFACE-WATER VOLUME (hm3):** This variable sums the volume of freshwater in the continental and coastal zones.

49 **FRESH GROUNDWATER VOLUME (hm3):** This variable sums the volume of groundwater in the continental and coastal zones.

50 **TOTAL FRESH-WATER VOLUMES (hm3):** This variable sums the volume of freshwater, both surface-water and groundwater, evaluated in the continental and coastal zones.

51 **TOTAL SALT-WATER VOLUMES (hm3):** This variable sums the volume of saltwater, both surface-water and groundwater, evaluated in the coastal zone.

52 **SURFACE-WATER RESERVE (hm3):** This variable sums the volume of surface-water, both freshwater and saltwater, evaluated in the

continental and coastal zones.

53 **GROUNDWATER RESERVE (hm3):** This variable sums the volume of groundwater, both freshwater and saltwater, evaluated in the continental and coastal zones.

54 **TOTAL WATER RESERVE (hm3):** This variable sums the volume of surface-water and groundwater, both freshwater and saltwater, evaluated in the continental and coastal zones.

3 Generate 1D Plots:

Plot Selection: Click this button to create the graph of selected global items.
Time: Select the units to represent the time in the graphs.
Compare: To generate the structure of a file .XLS with the measured data use the tool *MeasuredData* , in order to deal with files .XLS / .CSV of results. Select from the list the name of the section with measured data that you want to compare with the simulation results. The following describes the operation of the options in the drop-down list:

- **Do not compare**: empties the measured data that has previously been uploaded to the tool.
- **Add series ...**: loads the information from a *MeasuredData* file in .XLS / .CSV format. After importing the file, the names of the measured data series are added to the drop-down list, select a name from this list to compare with the model results.
- **Modify Time Scale**: modify the time scale of the measured data and transform it to daily, monthly, annual average values, or another unit of time, with the factor defined by the user. Selecting this option displays a window where you can configure the new time scale of the measured data. In the window that appears, activate or deactivate the transformation of the data using the activation box. The tool uses the interpolation method as *histogram with back step se* to make this modification.

Plot legend: legend that is displayed on the graph, modify the legend information for better identification. The legend does not work when multiple sections or items are selected.
Hold previous plot: Click on this check box to activate or deactivate the option to maintain the previous plot, once activated it is possible to maintain the previous graph to compare with the result of another simulation.
Mosaic of plots: Click on this check box to activate or deactivate multiple graphics, once activated it is possible to select more than one global item from the list.
Rows and Columns: These fields define the number of rows and columns of the graph mosaic that the tool should generate.

4 **XLS report:** export the selected global item in the tool list to an Excel file. In addition, it generates a graphic of the global item and store it as image format in a different sheet of the Excel file. If you need to export more than one global item, then activate *Mosaic of plots* to select more than one item and export them.

2.2.6.5.2 Flows DEB

Use this tool to analyze the file DEB with the flows simulated by the model through 1D graphs and descriptive statistical reports. In addition, it is possible to export the results in Excel format (XLS).

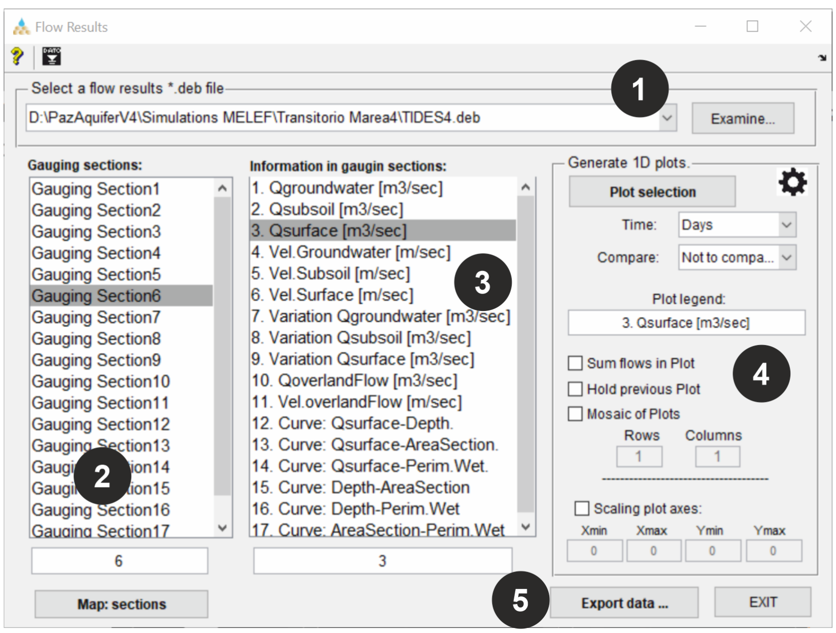


Figure 27: Tool to analyze the flows in the gauging sections of the model.

Usage of the tool sections marked with numeric labels in Figure 27:

1 **Select a flow results DEB file**

Examine: Select the file of flow results DEB to be analyzed.

Pop-up menu: This menu stores a list of the flow files you have analyzed, click on one of them to reload the information of that file.

2 **Gauging Sections**

List of gauging sections: select the section you want to analyze with one click over the section, or if you prefer, you can select several sections by activating the Mosaic of Plots.
Map of sections: Press this button to display a map showing the location of the different flow sections in the discrete domain.

3 **Information in Gauging Sections:** For a correct interpretation of the information of the flow sections, it must be taken into account that the negative or positive values of flow or velocity in a gauging section are mainly due to the direction that the flow takes; for example, a flow oriented downstream is positive whenever the gauging section is drawn from the right bank to the left bank of a river (counterclockwise) and vice versa; it must also be considered that the velocity field of surface water is evaluated for a triangular shape of the surface section encountered. Description of the list of information in a sections file:

1 **Qgroundwater [m3/seg]**: Groundwater flow through the gauging section.
2. **Qsubsoil [m3/seg]**: Subsoil water flow through the gauging section.
3. **Qsurface [m3/seg]**: Surface water flow through the gauging section.
4. **Vel.Groundwater [m/seg]**: Groundwater Darcy velocity.
5. **Vel.Subsoil [m/seg]**: Subsoil water Darcy velocity.
6. **Vel.Surface [m/seg]**: Surface water velocity.
7. **Variation Qgrounwater [m3/seg]**: variation of groundwater flow
8. **Variation Qsubsoil [m3/seg]**: variation of subsoil water flow.
9. **Variation Qsurface [m3/seg]**: variation of surface water flow.
10. **QoverlandFlow [m3/seg]**: Overland water flow through the gauging section.
11. **Vel.OverlandFlow [m/seg]**: Overland water velocity.
12. **Qsalt-groundwater [m3/seg]**: Salt groundwater flow through the gauging section. This parameter appears when the coastal model is active.
13. **Qsalt-subsoil [m3/seg]**: Salt subsoil flow through the gauging section. This parameter appears when the coastal model is active.
14. **Qsalt-surface [m3/seg]**: Salt surface flow through the gauging section. This parameter appears when the coastal model is active.
15. **Curve: Qsurface – Depth**: shows the rating curve Surface Flow – Water Depth.
16. **Curve: Qsurface – AreaSection**: shows the rating curve Surface Flow – Wet area.
17. **Curve: Qsurface – Perim.Wet**: shows the rating curve Surface Flow – Wet Perimeter.
18. **Curve: Depth – AreaSection**: shows the rating curve Water Depth – Wet area.

19.**Curve: Depth – Perim.Wet**: shows the rating curve Water Depth – Wet Perimeter.
20.**Curve: Area – Perim.Wet**: shows the rating curve Wet Area – Wet Perimeter.
All the items related with a rating curve (items 15 to 20) are assessed implementing the files of water levels VNO, nodal properties PRN and coordinates COR.

4 Generate 1D Plots:

Plot Selection: Press the button to graph the selected information.

: Configuration of lines and markers used in the flow graph to represent simulated or observed data.

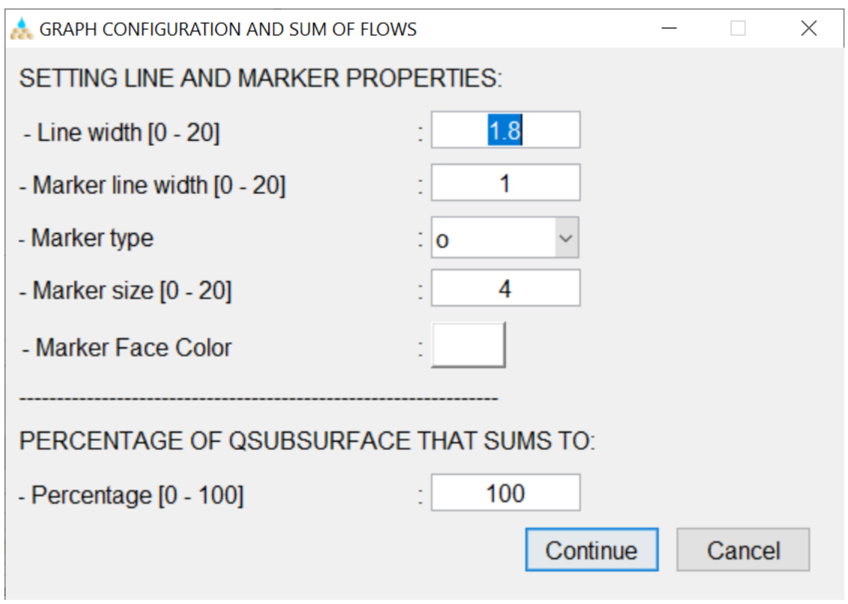


Figure 28: Settings of lines and markers of the tool Flows DEB. Furthermore, in this settings you can modify the percentage of Qsubalve that is added to Qgroundwater or Qsurfacewater.

- Line width [0 - 20]: Define the line width used to depict the hydrographs, velocities and flow variation.
- Marker line width [0 - 20]: Define the line width of the marker used to represent puntual values like observed or rate curves.
- Marker type: Select the marker type used to represent puntual values like observed or rate curves.
- Marker size: Define the marker size used to represent in the graph for puntual values.

- Marker Face Color: Click on the white box to change the color used as face color of the marker.
- Percentage [0 - 100]: Define the percentage of Qsubsurface flow that sums to Qsuperficial and Qgroundwater when this option (sum flows) is activated.

Time: Select a time unit from the pop-up menu list [Day, Hours, Minutes, Seconds] to display on the plot.
Compare: To generate a file with the measured data use the tool *MeasuredData*, in order to deal with files .XLS / .CSV of results. Select from the list the name of the section with measured data that you want to compare with the simulation results. The following describes the operation of the options in the drop-down list:

- **Do not compare**: empties the measured data that has previously been uploaded to the tool.
- **Add series ...**: loads the information from a *MeasuredData* file in .XLS / .CSV format. After importing the file, the names of the measured data series are added to the drop-down list, select a name from this list to compare with the model results.
- **Modify Time Scale**: modify the time scale of the data and transform it to daily, monthly, annual average values, or another unit of time, with the factor defined by the user. Selecting this option displays a window where you can configure the new time scale of the measured data. In the window that appears, activate or deactivate the transformation of the data using the activation box. The tool uses the step-back histogram interpolation method to make this modification.

Plot legend: legend that is displayed on the graph, modify the legend information for better identification. The legend does not work when multiple sections or items are selected.
Hold previous graph: check the box to overlap curves of different gauging sections on the same graph. Also use this option to load different simulation files and compare the flow rates from different simulations.
Mosaic of plots: Click on this check box to activate or deactivate multiple graphics, once activated it is possible to select more than one global item from the list.
Rows and Columns: These fields define the number of rows and columns of the graph mosaic that the tool should generate.
Graph scale (optional): check the box to define the limits of the X and Y axes of the plot or plot tile. Define the maximum and minimum values of each axis in order to to generate graphs with the same scale.

5 **Export data...**: Press this button to display Figure -2 where you can select the sections, the variables, the period, and the time increment of data to export.

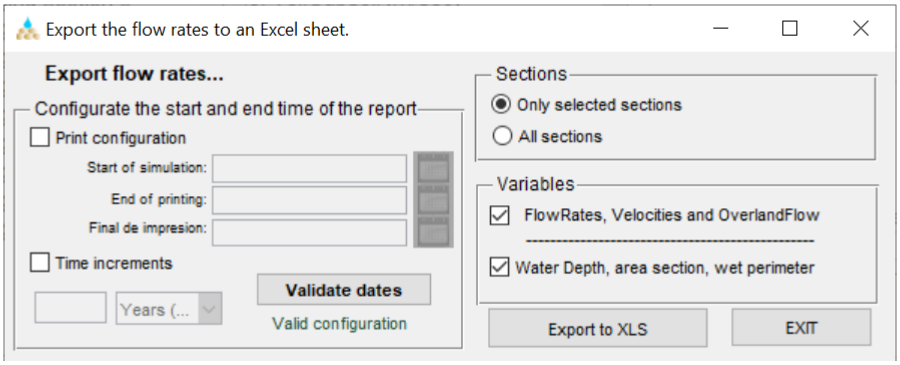


Figure 29: Tool to export flow rates of the DEB file.

Statistical Report of simulated Vs observed: Once the Flows DEB tool is configured, proceed to generate the graph with the hydrograph of the selected section and the respective measured values, if available. The Statistical option appears in the graph's menu bar to generate a summary of descriptive statistics, error, and numerical efficiency. For more information about the variables of this statistical report check the publication *Krause et al. 2005 (SRef-ID: 16807359/adgeo/2005-5-89).*

2.2.6.5.3 Nodes VNO-SOL-VEL

Use this tool to analyze the nodal evolution of the hydrological variables resulting from the model with the Nodal Results [VNO-SOL-VEL] tool.

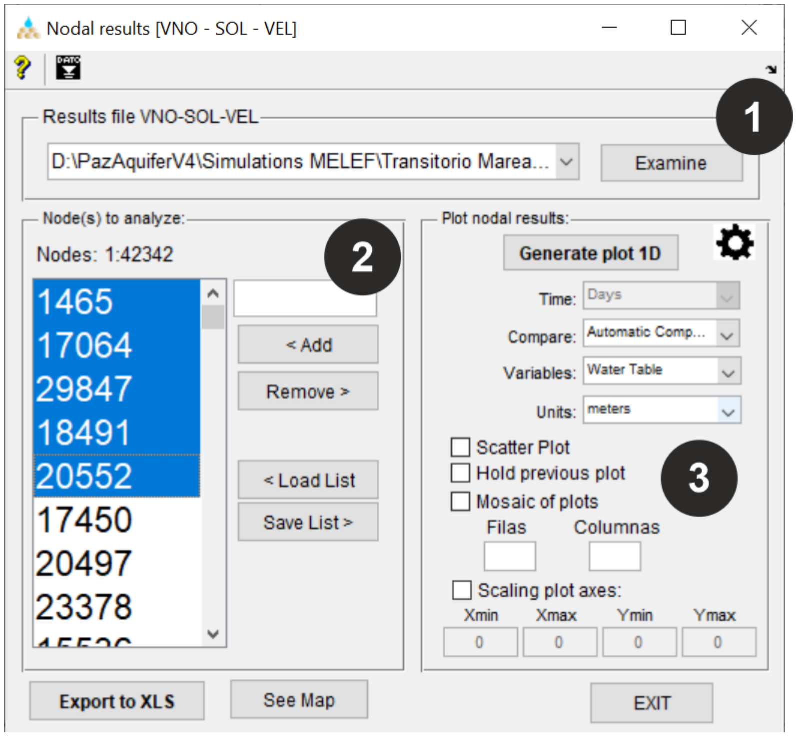


Figure 30: Nodal results tool for analysis of VNO-SOL-VEL simulation files.

Usage of the tool sections marked with numeric labels in Figure 30:

1 Results file VNO-SOL-VEL:

Examine: select a result file with extension VNO - VEL – SOL. The tool start indexing the result files for a quick reading and performs an automatic search for files with extension COR and ELE, applying a search criterion where these files must have the same name as the selected result file. If this criterion is not met, the multiple file selection window is displayed so that the user can apply a different criterion. These COR and ELE files are used to represent the results in 2D graphics, as well as to fix the EvapoTranspiration rates with the area of influence in each node.

2 Node(s) to analyze:

Write the number of node and add to the list to analyze different variables associated with the file of result selected.
<Add: Adds a node number to the node list, you can click this button or press enter.
Remove >: Removes the selected node number from the list.
<Load List: Load a list of nodes in CSV format, this list of nodes only has the number of nodes in a column with no text header.

Save List: Save the list of nodes that appears in the listbox in a CSV file format.
Export XLS: Exports the information of the nodes in the list to an Excel sheet.
See Map: Displays a map with the nodes of the mesh. At the bottom of the figure two controls allow you to modify the size of the nodes in the figure, and the number of contour lines according to the position of the scrollbar (min left = 0; max right = 200 contour lines). On the map, press the information button  to display the information window, then select any node to find out its elevation and node number (see Figure -1).

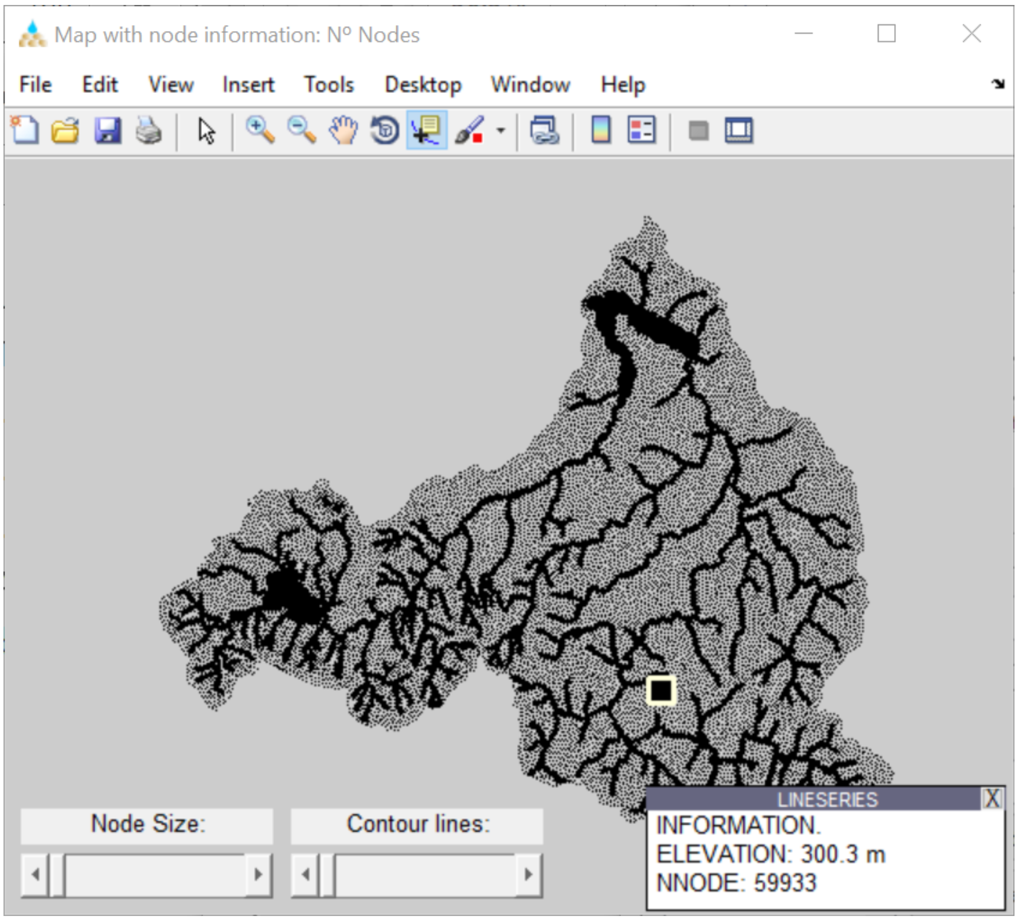


Figure 31: Map to identify the location, magnitud and number of the nodes, it also includes contour lines to identify areas by topography.

3 Plot nodal results:

Generate Plot 1D: Press this button to generate the 1D Plot of the nodes added to the list.
Time: Select a unit of time from the drop-down list [Days, Hours, Minutes, Seconds] to display on the graph.

Compare: To generate a file with the measured data use the tool *MeasuredData* , in order to deal with files .XLS / .CSV of results. Select from the list the name of the section with measured data that you want to compare with the simulation results. The following describes the operation of the options in the drop-down list:

- **Not to Compare**: Select this option to empty the measured data that has previously been uploaded to the tool.
- **Add Series ...**: Select this option to load the information from a *MeasuredData* file in .XLS / .CSV format. After importing the file, the names of the measured data series are added to the drop-down list, select a name from this list to compare with the model results.
- **Modify Time Scale**: modify the time scale of the data and transform it to daily, monthly, annual average values, or another unit of time, with the factor defined by the user. Selecting this option displays a window where you can configure the new time scale of the measured data. In the window that appears, activate or deactivate the transformation of the data using the activation box. The tool uses the step-back histogram interpolation method to make this modification.
- **Automatic Comparison**: Select this option to automatically load and compare the nodes associated with the measured location. After selecting this option a dialog appears to ask if you want to add the associated nodes to the list box of the tool. With this option enabled, select only the nodes and press *Generate plot 1D* to perform a comparison with the observed data of one or multiple nodes at a time.

Variables: select one of the variables from the list to represent their temporal evolution in a graph:

- **Water-Table**: This variable represent the elevation of the boundary between water-saturated aquifer and unsaturated aquifer.
- **Thickness Water Table**: This variable represents the saturated thickness of fresh water in the aquifer considering the substratum as starting point of this thickness.
- **Thickness Salt Water**: This variable represents the saturated thickness of salt water in the aquifer considering the substratum as starting point of this thickness.
- **Surface Water Depth**: This variable represents the thickness of surface fresh/salt water.
- **Pos. Fresh-Salt interface**: This variable represents the elevation at which the saltwater wedge is positioned in the aquifer.
- **Percentage Surface SaltWater**: This variable represents the percentage of salinity in the surface water. It uses the depth of sampling in the surface water

(by default it is the total depth, but the user can define it) and the thickness of the surface salt water to find the salinity percentage.

- **Percentage Ground SaltWater**: This variable represents the percentage of salinity in the groundwater. It uses the total saturated thickness of the aquifer and thickness saturated whit saltwater to obtain this percentage.
- **Depth to Water Table**: This variable represents the distance to the water table measured from the ground surface, or the difference between the topographic elevation and the water table elevation.
- **Depth to SaltWater Interface**: This variable represents the distance to the saltwater wedge position measured from the ground surface, or the difference between the topographic elevation and the elevation of the saltwater interface.
- **Water Table Change**: This variable represents the water table change between the time increments at which the variable is printed.

: Click this button to display the configuration window shown in Figure 32. In this configuration window it is possible to set the sampling depth (meters) of the surface water to evaluate the "Percentage Surface SaltWater", where by default a zero value is defined to consider the total depth of surface water. Enable or disable statistics [R2, RMSE, NASH-SUTCLIFFE GOODNESS OF FIT, BIAS] when preparing a comparison figure (observed Vs simulated). Finally, it is possible to set the date format displayed on the graphs.

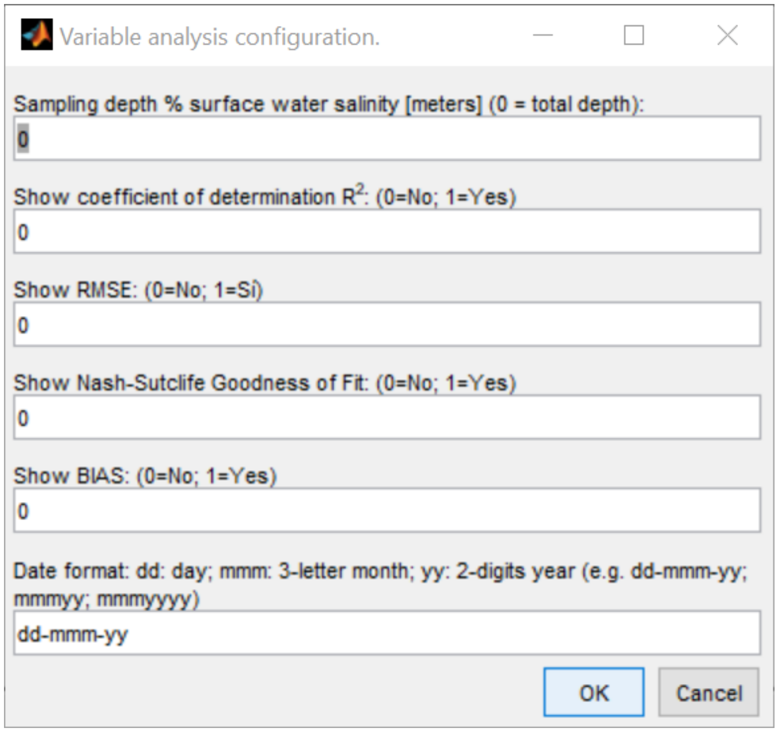


Figure 31: Configuration of parameters and statistics shown in figures.

Where the statistics implement the following equations:

STATISTIC	EQUATION	VARIABLES DESCRIPTION
Coefficient of determination (R^2)	$R^2 = 1 - \frac{RSS}{TSS}$	RSS: sum of squares of residuals TSS: total sum of squares. obs_i: Observed values. sim_i=simulated values.
Root Mean Square Error (RMSE)	$RMSE = \sqrt{\frac{\sum_{i=1}^{n} \left(obs_i - sim_i\right)^2}{n}}$	n = number of non-missing data points.
Nash Sutcliffe Goodnes of Fit (NSE-GOF)	$NSE - GOF = 1 - \left[\frac{1}{\left(\frac{SD}{RMSE} - 1\right) + 1}\right]^2$	SD: Standard deviation of the observed values. RMSE: Root Mean Square Error.
BIAS	$BIAS = \frac{\sum_{I=1}^{n} \left(sim_i - obs_i\right)}{\sum_{i=1}^{n} obs_i}$	

Units: select the units of the variable on the graph.

Hold previous plot: check the box to overlap curves of different gauging sections on the same graph. Also use this option to load different simulation files and compare the flow rates from different simulations.
Mosaic of plots: check the box to select multiple global items and tile charts. The mosaic is divided into n columns and n rows to represent the corresponding global items.
Scaling plot axes: check the box to define the limits of the X and Y axes of the chart or chart tile. Consider the following behavior:

- Define the maximum and minimum values of each axes in order to generate graphs with the same scale.
- When scaling is defined for time (X axes) the statistics use the data observed and simulated inside the defined period of analysis.
- Maintain the maximum and minimum value of the Y-axis to enable automatic scaling (requires defining the X-axis limits).

2.2.6.5.4 Zonal SOL

Use this tool to analyze the results printed in the SOL file in areas delimited by zones. To define the zones the tool requires the selection of a .dbf file of a shapefile layer with information in columns of [ZONA, AREA_M2, NNUDO] to process the results by zones.

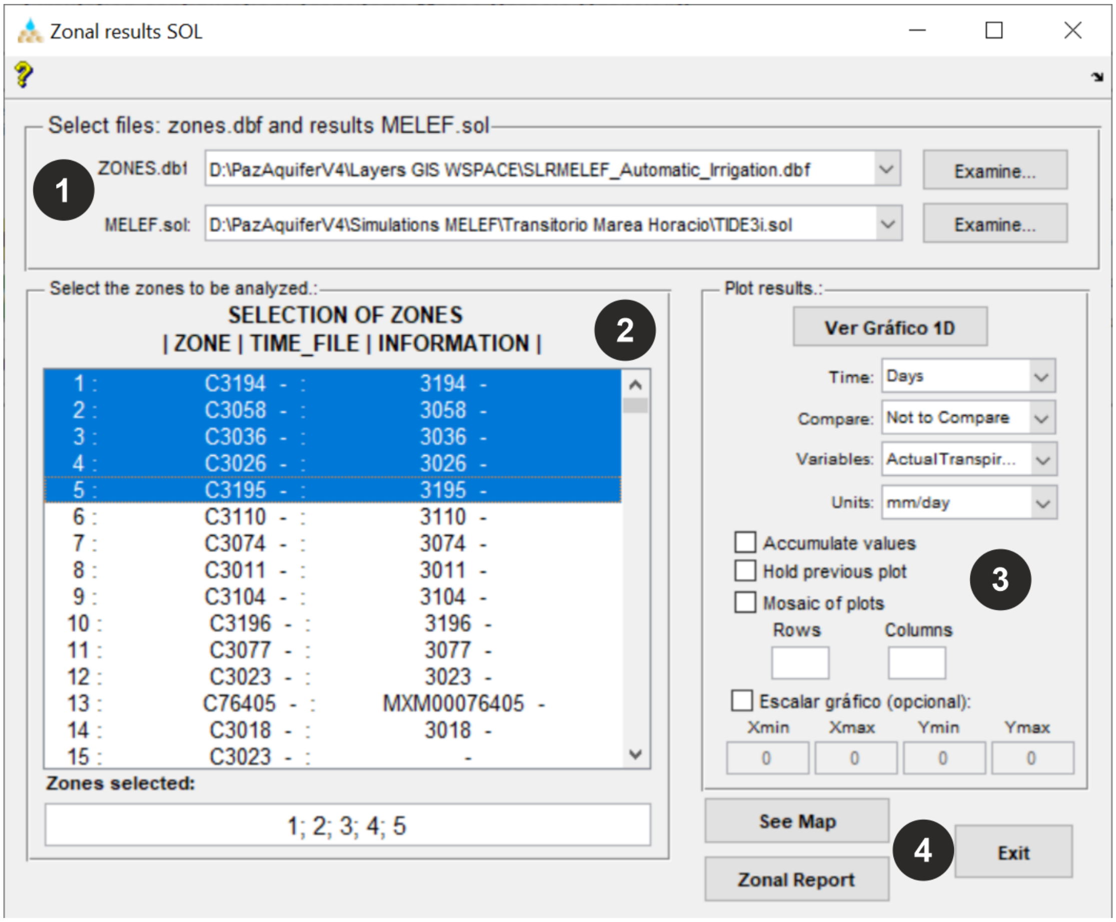


Figure 32: Tool for zonal analysis of SOL output files printed by MELEF FSW by zones defined in a shapefile layer (this tool uses the DBF file associated to the shapefile layer).

Usage of the tool sections marked with numeric labels in Figure 32:

1 Select files: zones.dbf and results MELEF.soi.

ZONES Examine: Select an attribute table (DBF file) from a point shapefile with simulation conditions generated with the GIS Toolbox (SLRMELEF, PRNMELEF, SOIMELEF). This layer may have the following fields in the attribute table [ZONA, AREA_M2, NNUDO], where the ZONA field defines the analysis zones, the AREA_M2 field defines the area of influence of each node and NNUDO is the node number assigned in the mesh.

MELEF.sol Examine: Select a .SOL result file generated by the MELEF FSW model for zonal analysis. NOTE: verify that the SLRMELEF and SOL files contain the same number of nodes (they belong to the same project).

2 **Select the zones to be analyzed**: Select the zones you want to analyze by graphing or printing in a report. Note that the reports will only contain the information of the previously selected or graphed zones.

3 **Plot nodal results**:
Generate Plot 1D: Press this button to generate the 1D Plot of the nodes added to the list.
Time: Select a unit of time from the drop-down list [Days, Hours, Minutes, Seconds] to display on the graph.
Compare: To generate a file with the measured data use the tool *MeasuredData*, in order to deal with files .XLS / .CSV of results. Select from the list the name of the section with measured data that you want to compare with the simulation results. The following describes the operation of the options in the drop-down list:

- **No to compare**: Select this option to avoid any comparison or stop comparing in case there are measured data that have been previously loaded into the tool.
- **Add Series...**: Select this option to load the information from a *MeasuredData* file in .XLS / .CSV format. After importing the file, the names of the measured data series are added to the drop-down list, select a name from this list to compare with the model results. Time
- **Modify Time Scale**: Modify the time scale of the measured data and transform it to daily, monthly, annual average values, or another unit of time, with the factor defined by the user. Selecting this option displays a window where you can configure the new time scale of the measured data. In the window that appears, activate or deactivate the transformation of the data using the activation box. the tool uses the step-back histogram interpolation method to make this modification.
- Select this option to load the information from a *MeasuredData* file in .XLS / .CSV format. After importing the file, the names of the measured data series are added to the drop-down list, select a name from this list to compare with the model results.

Variables: Select the variable you want to plot on the graph.
Units: Select the units of the variable on the graph.
Hold previous plot: Check the box to maintain the plotted lines of different zones. Also use this option to plot results from different simulation files and compare the results in the same graph.
Mosaic of plots: Check the box to be able to generate a mosaic of graphics. The graph mosaic is divided into m columns and n rows to generate a total of m x n graphs in the mosaic.

Scaling plot axes: Check the box to define the limits of the X and Y axes of the graph or mosaic graphs. Define the maximum and minimum values of each axes in order to generate graphs with the same scale.

4

See Map: Displays a map with the nodes of the mesh. At the bottom left of the figure three controls allow you to modify the size of the nodes in the figure, modify the number of contour lines according to the position of the scrollbar (min left = 0; max right = 200 contour lines) and search for the zones of the SLRMELEF file (see the Figure 33). On the map, press the information button and click on nodes to display information about the elevation and node number.

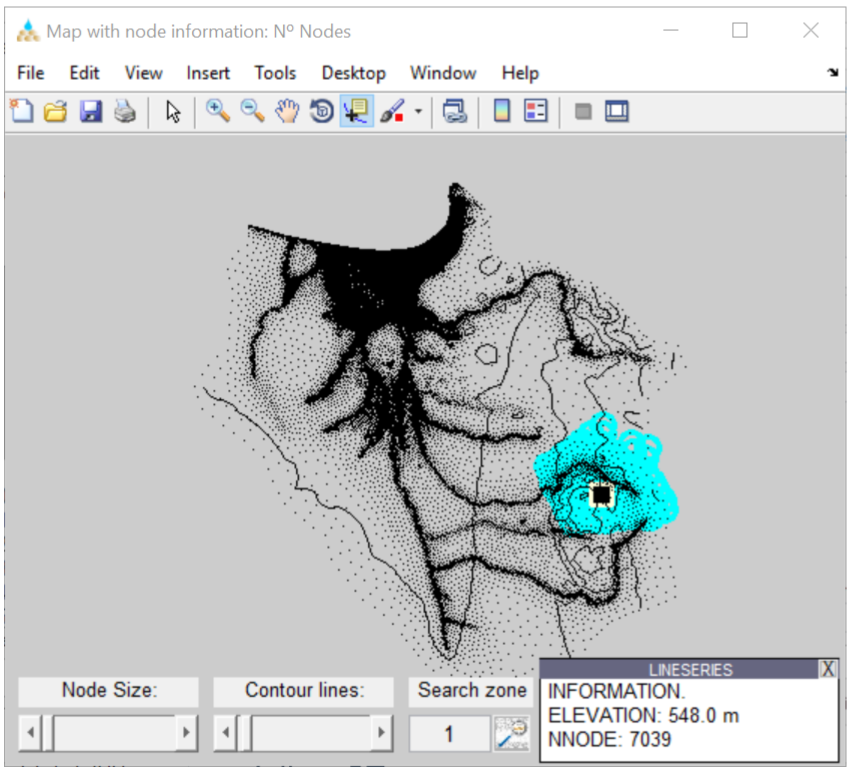


Figure 33: Map displayed to identify nodal properties and find out zones.

Zonal Report: Exports the information of the selected or previously graphed zones in .XLS or .CSV format.

2.2.6.5.5 Animations 1D VNO-VEL

1D: VNO-VEL

Use this tool to create maps of phreatic water levels and velocities and analyze the evolution of the phreatic levels by means of section views following any given route. In addition, you can also draw an area where you can observe the velocities by means of vectors. In both cases, it is possible to generate animations of the temporal evolution of the simulated variables.

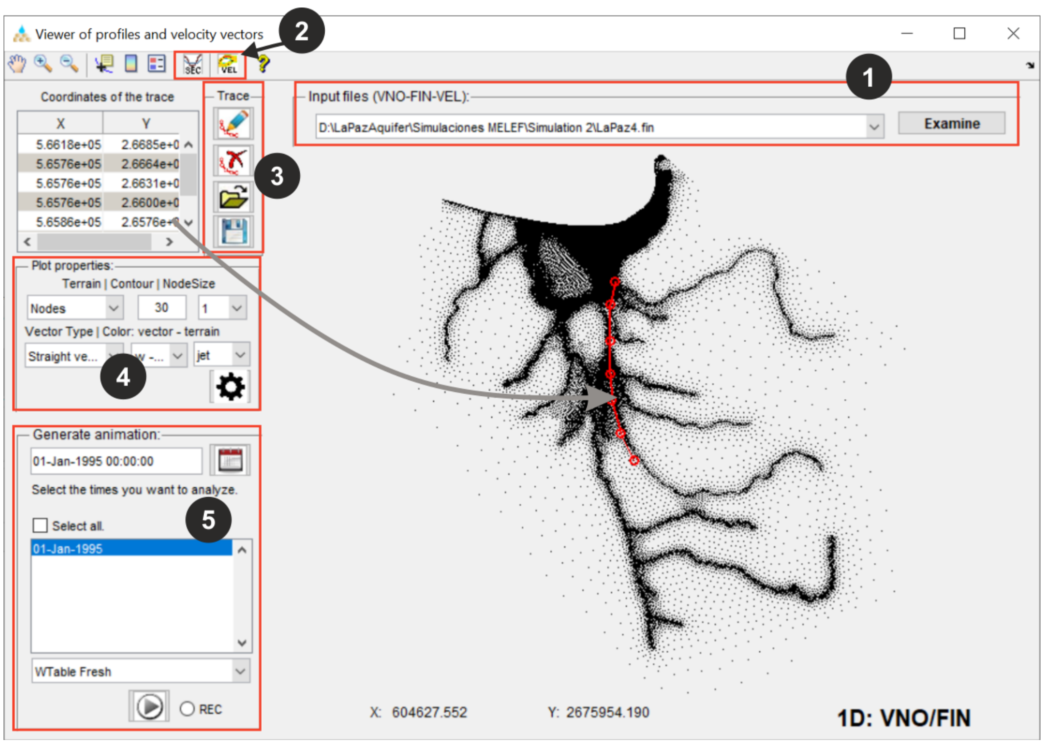


Figure 34: Results visualization tool, this tool creates hydrological vertical profiles and maps of velocity vectors.

Usage of the tool sections marked with numeric labels in Figure 34:

1 INPUT FILES (VNO-VEL):

Examine ...: Press this button to select a result file with extension .VNO - .VEL - .SOL. The tool indexes the files for quick reading, and performs an automatic search for files with extension .COR, ELE. and .PRN, applying a search criterion where these files must have the same name as the selected result file. If this criterion is not met, the multiple file selection window is displayed so that the user can apply a different criterion.

Drop-down menu: this drop-down menu stores the paths to the results files you have analyzed in the current session; click on one of the paths in the drop-down menu to reload the results that correspond.

2 Select the graph to be generated:

Use both buttons on the toolbar to activate the different chart options:

Section view: modifies the editing pencil in section 3 to draw a path for viewing the vertical profile, or vertical section view, of the water table, topography, and impervious substrate.

Velocity vectors: modify the editing pencil in section 3 to draw an area in which to observe the velocity vectors, modulus, and direction of the subsurface - sub-surface - surface velocity, with the contour lines of the topography as background; a .VEL file must be loaded in order to use this option.

3 Editing Toolbar: Trace

Plot and manage with this toolbar the route to be followed by the vertical profile graph of the water table, or the analysis area of the velocity vectors.

Drawing a Polyline:

- Finish Editing: press the right mouse button.
- Continue previous trace: press the editing pen again.
- Modify vertices: only possible by modifying the XY coordinates in the table.

Drawing a polygon:

- Finish Editing: press the right mouse button.
- Insert vertices: press the A key + mouse click on an edge of the polygon.
- Modify vertices: drag the circle of the vertex with the mouse.

Delete Trace: press this button to delete the trace that is currently active.

Open Trace: Press this button to load a trace from a .S1D file extension.

Save Trace: press this button to save the current trace in a .S1D file extension.

4 Plot properties:

Modify visual settings to improve analysis of results.

Drop-down menus:

- **Terrain|Contour|NodeSize:** visualize the surface with one of the available options [Nodes; Terrain; Contour Lines]
  - **Nodes:** This option displays the nodes of the mesh, use the pop-up menu *NodeSize* to change the nodes size.
  - **Terrain:** display a continuous terrain with a jet palette color by default.
  - **Contour:** this option displays the contour lines for the terrain, specify the number of contour lines you wish to observe in *Contour* text box, the default is 60.
- **Vector Type| Color: vector - terrain:** Configuration of the type and color of vector used with velocities, and the palette color of terrain.
  - **Straight vector - solid color:** Select this option to generate straight vectors with solid color.
  - **Curved vector - graduated color:** Select this option to generate curved vectors with a graduated color in function of their magnitude.
  - **Color vector:** Select a color from the list.
  - **Terrain color:** Select a palette color from the list.

Set vector configuration: define other configurations for vectors

- **Straight vector:**
  - **Vector size:** define the scaling factor of the size applied to the original vector (by default = 2)
  - **Position of the scale legend:** four possible positions [1-4], where 1 defines the upper right position, 2 the upper left position and so on (by default =1)
  - **Units shown in the legend:** Change the legend text of the vector in the chart.
  - **Conversion factor for units:** change the units using the required conversion factor, e.g. conversion factor 86400 converts from m/sec to m/day (by default = 1)
- **Curved vector - graduated color:**
  - **Maximum vector length:** this value is a factor that multiplies the average distance of the nodes within the plotted area.
  - **Vector density factor:** This is an inverse factor, so that if it is greater than one the density decreases, and if it is less than one the density of displayed vectors increases. Note that the higher the vector density, the higher the memory consumption of the computer and the longer you will have to wait to see the result.
  - **Vector thickness:** Define the thickness of the vector line.

- **Units shown in the legend:** Change the legend text of the vector in the chart.
- **Position of the scale legend:** Numerical value from 1 to 4, where 1 positions the legend at the top right, value 2 at the top left, and so on.
- **Minimum vector magnitude:** set the lower limit of the vector magnitude you want to display.
- **Maximum vector magnitude:** set the higher limit of the vector magnitude you want to display.
- **Conversion factor for units:** change the units using the required conversion factor, e.g. conversion factor 86400 converts from m/sec to m/day (by default = 1)

5 **Generate animation:**

 **Calendar:** The tool automatically fills it with the start date and time defined in the project. You can click on calender to define a new start date and time or type the date directly into the text box on the left of the calendar.

List of dates: Select one or more dates to display on the plot, note the following ways to select the dates:

- Drag the mouse over the list while holding down the left mouse button.
- Use Ctrl + click to select non-consecutive dates.
- Use Shift + click ti select all dates between two different dates.
- Check the **Select All** box to select all dates.

Variables to Plot:

- **WTable Fresh**: The profile 1D represents only fresh water level.
- **WTable Fresh/Salt**: The profile 1D represents the fresh and saltwater levels.

 **Run animation:** Press the button to view the graph of the selected dates.

 **REC:** Press this button to activate the option to record a *video*.

NOTE: before running an animation, it may be useful to press play by selecting only one date, zoom in on an area of interest and without closing the chart return to the main window to select multiple dates and press play again.

2.2.6.5.6 Animations 2D VNO-SOL-VEL

2D: VNO-VEL

Use this tool to generate graphs with animation of the 2D results: VEL - VNO - SOL. This tool allows to visualize all the variables that are printed in the result files of velocities, groundwater levels and position of the fresh/salt interface, as well as the climatic variables of evaporation, evapotranspiration and the different water uses evaluated internally by the model.

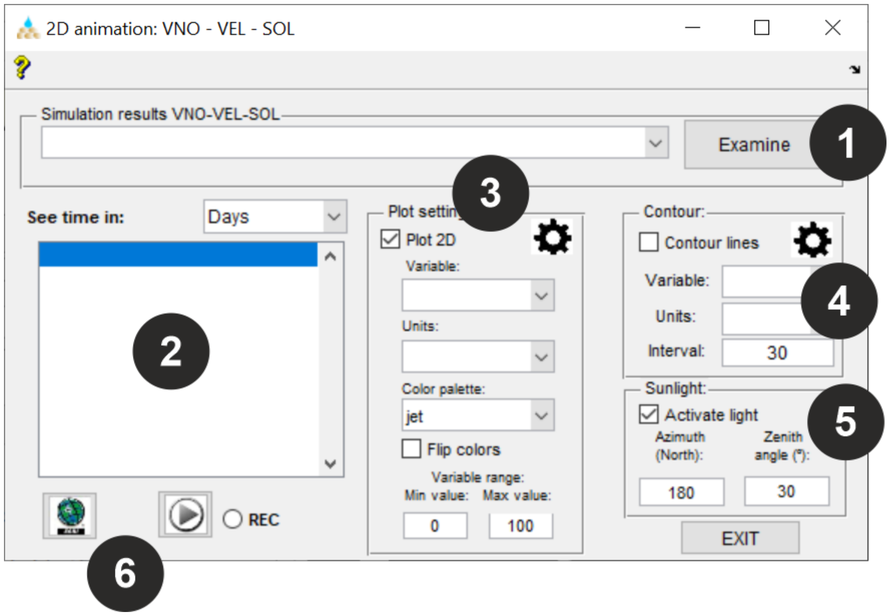


Figure 35: Tool to generate videos with results 2D: VNO - SOL

Usage of the tool sections marked with numeric labels in Figure 35:

1 **Examine:** Select a result file with extension VNO, VEL or SOL. The tool performs an automatic search for files with extension COR and ELE, and if this is not met the *multiple file selection* window is displayed for the user to apply a different criterion. The periods with results are added to the list in section with label 2.

2 **Select the time increments to display:** select one or more time increments dragging the mouse over the list while holding down the left mouse button, or use Ctrl + click to select non-consecutive time increments. NOTE: before running an animation, it may be useful to press play by selecting only one date, zoom in on an area of interest and without closing the chart return to the main window to select multiple dates and press play again.

Time pop-up: select the increment of time to represent the periods.

3 **Plot settings:**

Plot 2D: Uncheck this box if you do not want to see the graph 2D, as you may want to see only the contour lines of this variable.

Variable: Depending on the type of result files you select [VNO, VEL, SOL], the different variables will be loaded in this drop-down menu, select the variable you want to see in the graph.
From VNO file you can select the following variables:

- **Water-Table**: This variable represent the elevation of the boundary between water-saturated aquifer and unsaturated aquifer.
- **Thickness Water Table**: This variable represents the saturated thickness of fresh water in the aquifer considering the substratum as starting point of this thickness.
- **Thickness Salt Water**: This variable represents the saturated thickness of salt water in the aquifer considering the substratum as starting point of this thickness.
- **Depth Surface Water**: This variable represents the thickness of surface fresh/salt water.
- **Pos. Interface Salt-Fresh**: This variable represents the elevation at which the saltwater wedge is positioned in the aquifer.
- **Percentage Surface SaltWater**: This variable represents the percentage of salinity in the surface water. It uses the depth of sampling in the surface water (by default it is the total depth, but the user can define it) and the thickness of the surface salt water to find the salinity percentage.
- **Percentage Ground SaltWater**: This variable represents the percentage of salinity in the groundwater. It uses the total saturated thickness of the aquifer and thickness saturated whit saltwater to obtain this percentage.
- **Depth to Water Table**: This variable represents the distance to the water table measured from the ground surface, or the difference between the topographic elevation and the water table elevation.
- **Depth to SaltWater Interface**: This variable represents the distance to the saltwater wedge position measured from the ground surface, or the difference between the topographic elevation and the elevation of the saltwater interface.
- **Water Table Change**: This variable represents the water table change between two time increments.

: Configuration of the sampling depth (meters) of the surface water to evaluate the "Percentage Surface SaltWater". By default a zero value is defined to consider the total depth of surface water.
Units: Depending on the variable you select; this drop-down menu will display the units of measurement you can choose from.
Color Palette: Select the color palette that best suits your needs.

Variable range: The range of the variable allows you to adjust the color palette to the maximum and minimum expected value.

4 **Contour:**

Contour lines: Uncheck this box if you do not want to see contour lines in the 2D graph (map).
Variable: Define the variable you want to use to represent their contour lines.
Interval: Define the number of intervals required for the contour lines.
: Configuration of label properties like activate it, spacing between labels, label size and color; contour lines properties like base value, color, size and line style; or activation/deactivation of catchment line.

5 **Sunlight:**

Activate light: Uncheck this box if you do not want to see the effect of lights on the map.
Azimuth (North): Define the direction from which the light is generated in the graph.
Zenith angle(º): Define the light zenith angle (sun's rays) in degrees.

6 **Plot/Export selected periods:**

Run animation: Press the button to view the graph of the selected dates.
REC: Press this button to activate the option to record a *video*.
Export to ASCII for GIS: This button export an ASCII file for each period selected and can be imported from ArcMap directly to create a shapefile of points whit all the columns of information of the results file analyzed. The process to import this ASCII file to ArcMap is:
1) Open ArcMap and go to *Catalog Window*
2) Find the path where the ASCII file is and right click to it and select the option *Create Feature Class>From XY Table ...*
3) Follow the instructions to select the columns with coordinates and import the file to create a shapefile of nodes.
NOTE: this button also creates another file called Schema.ini which is required by ArcMap to identify the text delimiter used in the ASCII file.

2.2.6.5.7 Zonal Balance

Use this tool to generate a water balance in areas completely delimited by gauging sections. The tool requires the selection of multiple files to analyze all water inputs and outputs to the area of analysis.

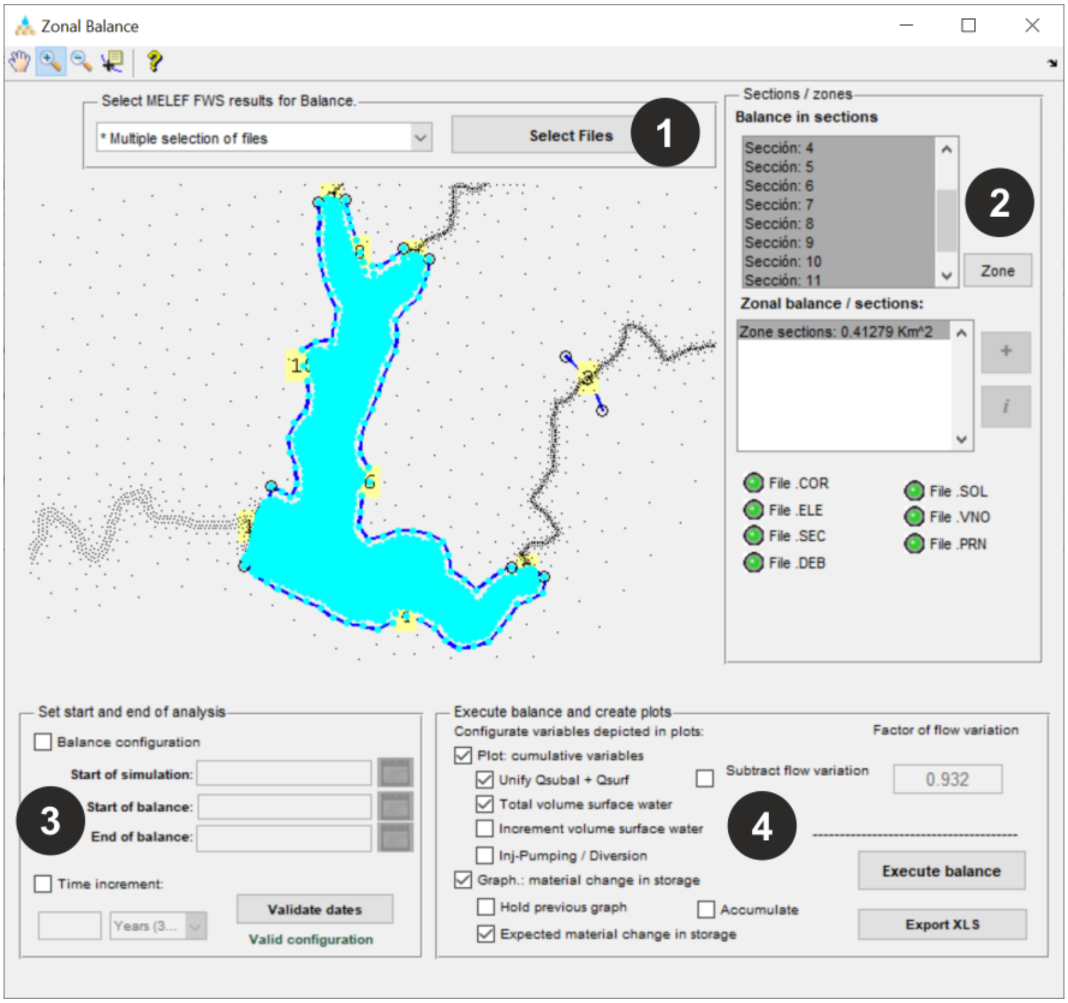


Figure 36: Tool to execute balances in areas completely delimited by gauging sections. This tool requires the selection of multiple files for analysis.

Usage of the tool sections marked with numeric labels in Figure 36:

1 **Select MELEF FSW results for Balance:**

Click the *Select Files* button to display a dialog window that notifies the user which files are required for the water balance, these files must be selected from the window *multiple file selection*, see Figure 37; remember that these files must belong to the same simulation. After selecting these files the tool draws the nodes and the gauging sections in the main window, use the toolbar on the top left to zoom in or out the graph.

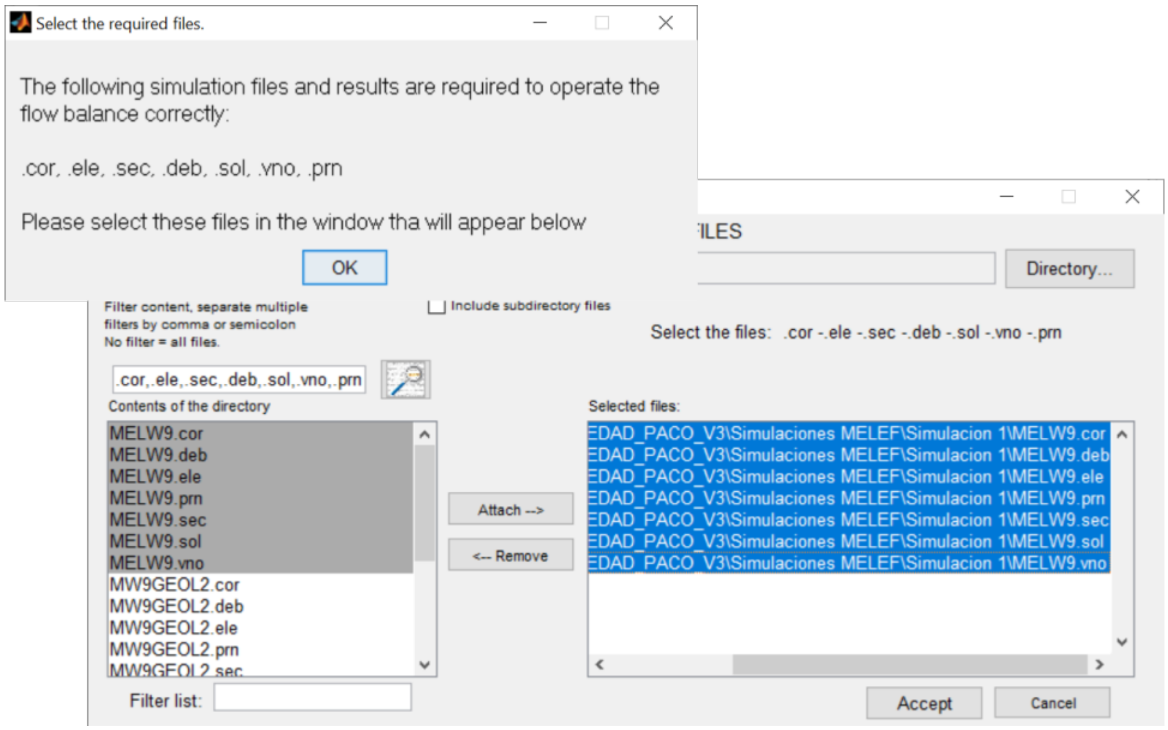


Figure 37: Select the required files in the multiple file selection window.

2 Sections / Zones:

After load the required files [COR, ELE, SEC, DEB, SOL, VNO, PRN] the list of sections appears. Select the sections that enclose the area of balance and click the button *Zone*. To find out which sections meet this requirement, zoom in toolbar and look at the sections numbers, which are highlighted with a yellow background color. If the sections do not completely delimit the balance area, or in other words the sections do not form a closed zone, then pressing the Zone button will not highlight the nodes of the analysis area in cyan color and the analysis area will not appear in the list of the *Zonal Balance* section.

What performs the button *Zone*:

- Evaluates whether the sections form a closed zone.
- If this is the first time you are doing a zonal balance in the current directory, then the tool evaluates the area of influence of the nodes in the file areanodos.nod, as well as the area of each triangular element in the file areaelementos.tri. Both files are reused in future balance analysis.

- It searches for nodes that are located within the area delimited by the gauge sections and highlights them with cyan color. It also adds, in the box below the *Zone* button, an item with information on the area of the analysis zone.
- It deletes the results of a previous balance from the memory and allows to run the analysis of a new balance.

3 Set start and end of analysis:

Balance configuration: check this box if you need to perform a balance in a particular time window setting the start date and end date of the balance, use the calendar or type manually the dates of analysis. The other option is maintain uncheck the box to generate a balance for the whole printed information in the results files using as a time stamp the days of simulation.

Time increments: you can modify the time increment of the displayed balance, daily, weekly, biweekly, monthly, yearly, using for this the interpolation as histogram with backward step, independently of the printing frequency in the files. Monthly balances, if you activate the start and end dates of the balance the the tool can know exactly the duration of each month, if not, the duration of a month is considered equal to 30.14375 days (considering the leap year).

Validate dates: It is required to press this button to validate the configuration of dates and time increment, without validate the configuration then the water balance can't be executed.

4 Executing balance and create plots:

Execute balance: press the button *Execute balance* to start the balance and perform the following:

1. Performs a total water balance including the following: groundwater, sub-surface water and surface water flows that input or output the analysis area; if the simulation considers the continental and coastal coupled model (AL2S model) then saltwater is included as groundwater.salt, subSurfaceWater.Salt and SurfaceWater.Salt ; the outputs by water evaporation and transpiration; the water uses represented by pumping / injection / bypassing of water if it were defined.
2. In the first graph it shows a graph with the evolution of all inputs and outputs in the analysis area, see Figure 38. Also includes the evolution of the surface water storage and changes of surface water in the analysis area, which in comparison with the total balance allows to deduce the storage in the

materials, or to show an incorrect balance due to the lack of calibration. See the following list of options you can enable or disable to see in this graph:

a. **Plot: cumulative variables:** uncheck this box to avoid display the graph with the evolution of water inputs and outputs.
b. **Unify Qsubal + Qsuperf:** check this box to sum the flows of sub-surface flow and surface flow and show only one dashed line with the evolution of this two variables. In case there are saltwater flows it unify the flows of Qsubal.Salt and Qsuperf.Salt in only one dotted line.
c. **Total volume surface water:** check this box to show the volume of surface water storage simulated in the analysis area.
d. **Increment volume surface water:** check this box to show the volume of surface water storage as increment.
e. **Inj-Pumping / Diversion:** Check this box to show the water volume of pumping - injection - diversion.
f. **Subtract flow variation:** Check this box to subtract the flow variation of groundwater - subsurface water - surface water flows. The flow variation is an approximation of the uncertainty of the gauging section calculation methodology. Use the factor at the right of this option to define if the flow variation should be added or subtracted [-1 to 1]. In case there is freshwater and saltwater flows, then this variation is subtracted from each by weighting by the percentage of freshwater and saltwater flowing through the gauging section.

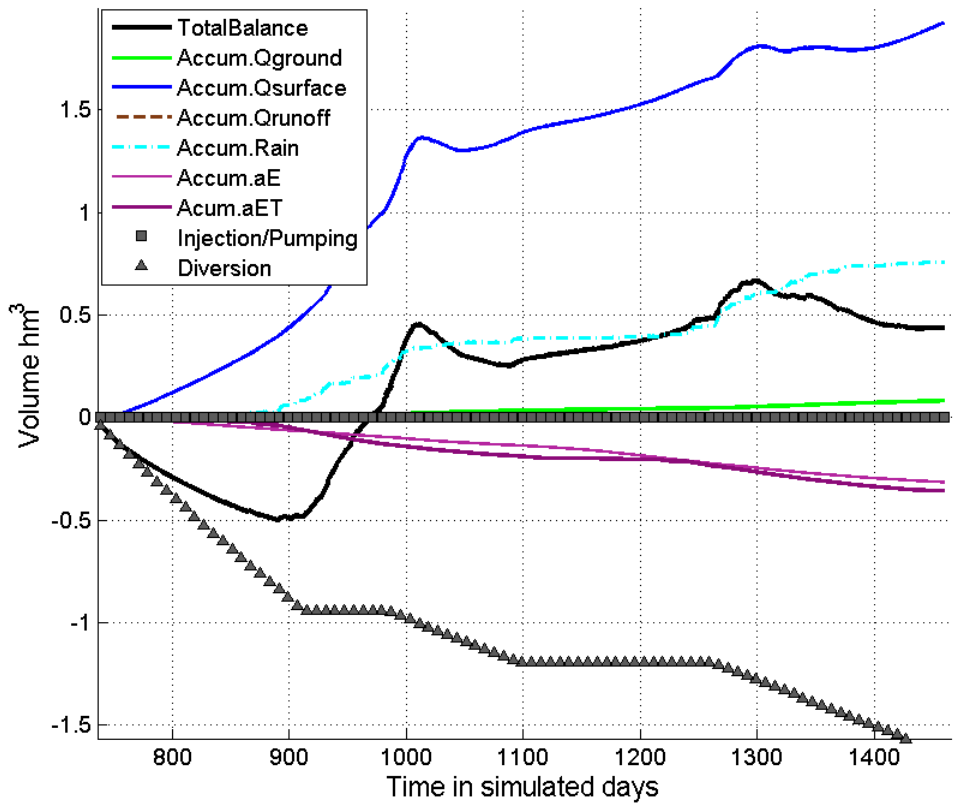


Figure 38: Graph with the water balance performed by the tool Zonal Balance.

3. In a second graph it shows the expected evolution of the change in storage in materials, as well as the change in storage obtained indirectly by subtracting the accumulated volume of the total balance and the accumulated volume of surface water, see Figure 39. A good calibration will show a similar behavior of both curves. See the following list of options you can enable or disable to see in this graph:
   a. **Plot: material change in storage:** uncheck this box to avoid display the graph with the change in storage evaluated by balance and expected by the groundwater level change.
   b. **Hold previous graph:** check this box to maintain a previous graph and compare with another simulation.
   c. **Expected material change in storage:** uncheck this box to avoid display the evolution of the change in storage expected by the groundwater level change.
   d. **Accumulate:** check this box to show the cumulative evolution of the change in storage.

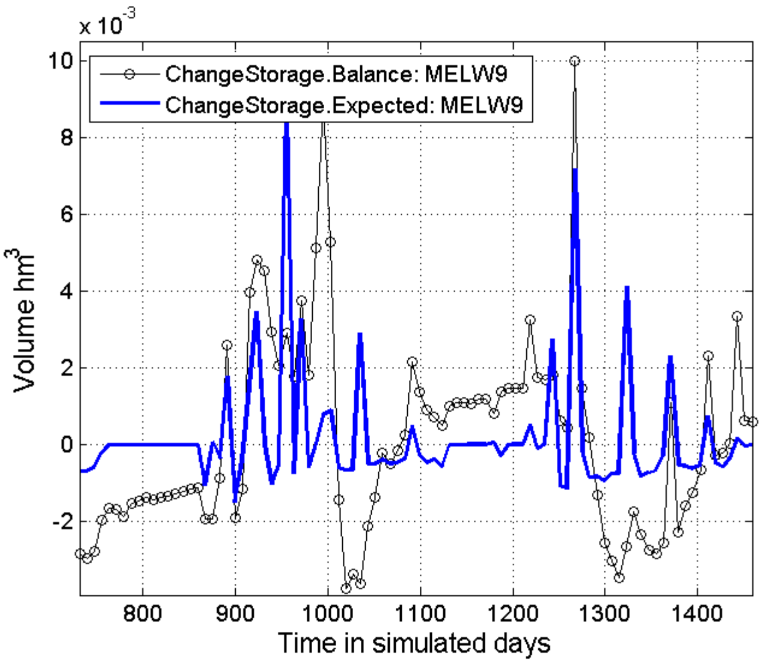


Figure 39: Graph with the materials change in storage evaluated by Balance and expected by groundwater levels in materials.

4. The balance results is stored in temporal memory, so you can activate or deactivate the balance configuration boxes and press the button *Execute balance* again to see the changes instantly.

Export XLS:

- Export to an Excel file the variables involved in the zonal balance.
- On the same Excel file, but in a different sheet, it export the values of the volume/increase of surface water volume.

2.3 Toolbox GIS

2.3.1 1. Import Mesh

2.3.1.1 1.1 Import Nodes - Mesh (.GEO - .2DM - .MSH)

Summary

This tool import .GEO, .2DM format files of the AQUAVEO SMS SOFTWARE, and the .MSH format file of the IBER software, to create shapefiles of nodes, triangular elements and catchment polygon area. These files are necessary to manage all properties and simulation conditions of the MELEF FSW model.

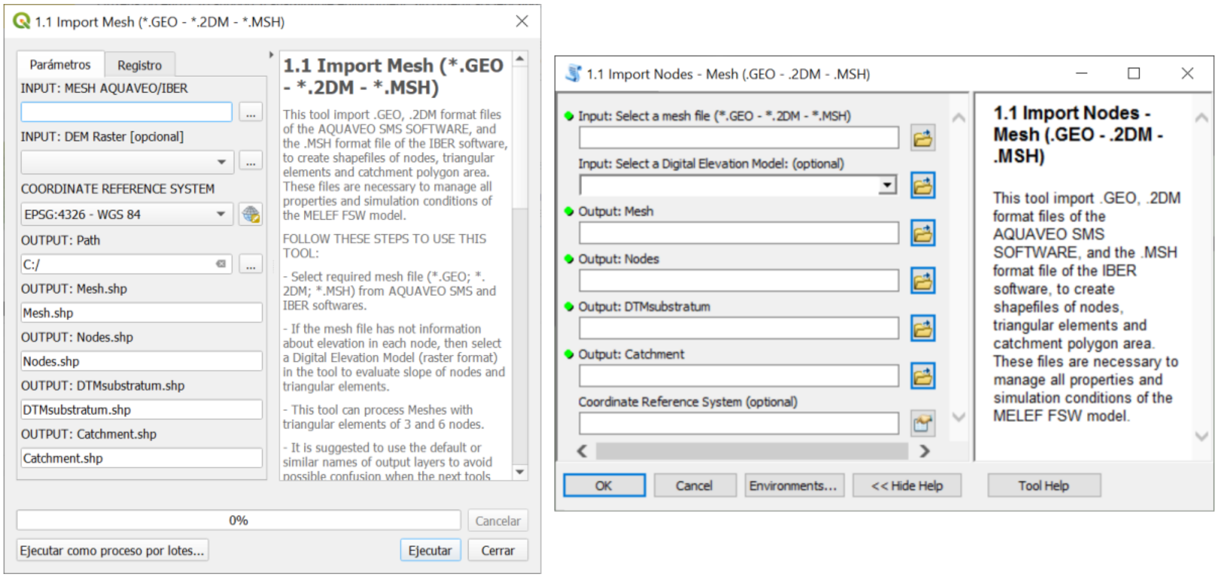


Figure 40: Tools to import the mesh in QGIS (Left) and ArcMap (Right).

Usage of the tool (Figure 40)

FOLLOW THESE STEPS TO USE THIS TOOL:
- Select required mesh file (*.GEO; *.2DM; *.MSH) from AQUAVEO SMS and IBER software's.
- If the mesh file has not information about elevation in each node, then select a Digital Elevation Model (raster format) in the tool to evaluate slope of nodes and triangular elements.
- This tool can process meshes with triangular elements of 3 and 6 nodes.
- ArcMap: Before run the tool go to Geoprocessing>Environments...>Workspace and define the Workspace and ScratchWorkspace paths to the project paths WSPACE and SWSPACE generated by MELEF FSW Interface.
- It is suggested to use the default or similar names of output layers and avoid possible confusion when the next tools ask for these layers as inputs.

OUTPUT LAYERS:
- Nodes.shp: point layer with the nodes of the mesh.
- Mesh.shp: polygon layer with the triangular elements of the mesh.
- DTMsubstratum.shp: point layer with the nodes of the mesh and parameters to define the elevation of the Digital Terrain Model and location and the impervious substratum.
- Catchment.shp: polygon layer with the limits of the analysis area defined by the triangular elements.

ATTRIBUTE TABLE VARIABLES:
- NNUDO: Numbering of nodes defined in the mesh file.

- ELEMENTO: Numbering of triangular elements defined in the mesh file.
- SLOPE: Slope in degrees of triangular elements of the mesh | slope of nodes evaluated as the average slope of all triangular elements related with each node.
- AREA_M2: area of influence of each node in the mesh.
- POINT_X: Coordinate X of nodes.
- POINT_Y: Coordinate Y of nodes.
- POINT_Z: Elevation used to evaluate the slope of nodes and elements.
- SUSTRATO: empty field required to save the Coordinate Z of the impervious substratum.
- COTA1: empty field required to save the Coordinate Z of the nodes.
- COTA2: empty field required to save secondary elevation of nodes to define the presence of Tunnels that is activated when COTA2 is lower than COTA1.
- WEATHERING: Difference between COTA1 and SUSTRATO.

NOTE: When the mesh file has not information of elevation you need to use an external source of elevation like a Digital Elevation Model in a raster format. The slope evaluation is performed in degrees for each triangular element of the mesh, meanwhile the node slope is evaluated as the average slope of all triangular elements related with each node.

2.3.2 2. Non Saturated Zone - EvapoTranspiration (SOI)

2.3.2.1 2.1 New Non Saturated Zone

Summary

This tool generates a new (empty) shapefile of polygons and defines the structure of the attribute table. This layer must be edited to define the polygons of the different non saturated zones, if polygons already exist in other layers, then you can copy existent polygons and paste into this layer.

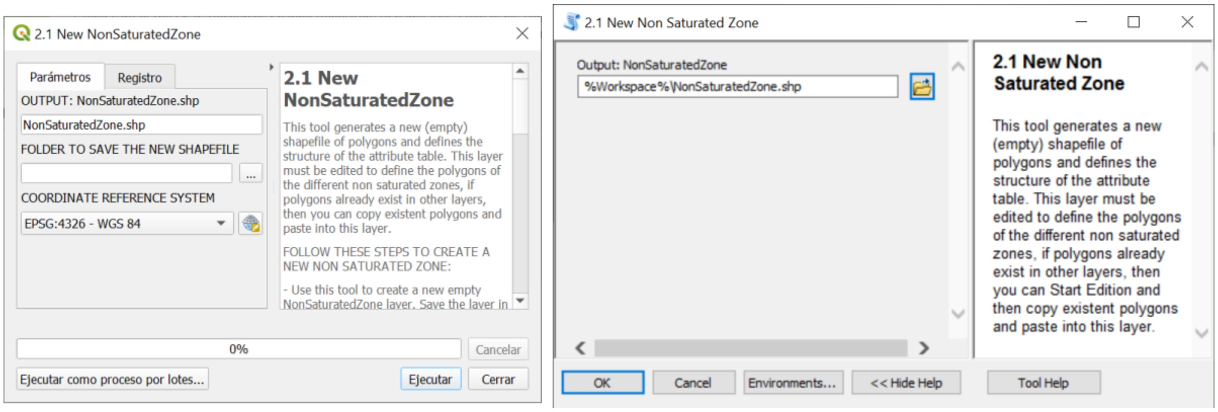


Figure 41: Tools to create a new empty layer NonSaturatedZone with the predefined attribtue table structure in QGIS and ArcMap.

Usage of the tool (Figure 41)

FOLLOW THESE STEPS TO CREATE A NEW NON SATURATED ZONE:

- Use this tool to create a new empty NonSaturatedZone layer. Save the layer in the project workspace created by the MELEF FWS model.
- Edit the layer to define different polygons related with distinct soil types or land cover: you can copy polygons from other layers (ArcMap: Start Edition and copy, ctrl-c, the polygons from other source layers and paste, ctrl-v, to the target layer).
- Fill with values the parameters in the attribute table.

ATTRIBUTE TABLE DESCRIPTION:

- EC: Capillary Fringe Thickness [Meters]
- ES: Soil Thickness related with the maximum depth of vegetation roots [Meters].
- CCAMPO_PM: Percentage of Readily Available Water in soil for EvapoTranspiration [%].
- ETP: Potential EvapoTranspiration. The units can be defined by user in the field [UNIDADES](mm/day, mm/día, check manual for more units), or use default units (meters/second) if UNIDADES is empty.
- EP: Potential Evaporation from surface water, see description of ETP for units.
- UNIDADES: define the units of ETP and EP, if this field is empty then default units are considered (meters/second).
- ZONA: Unique identification of each polygon, this field is managed automatically.
- INFO: field for general purpose information to describe each polygon.

- TIME_FILE: define here the name of the "COLUMN" in an Excel file where there is information about the transient behavior of fields [EC, ES; CCAMPO_PM; ETP; EP]

2.3.2.2 2.2 New Ksuperficial

Summary

This tool generates a new (empty) shapefile of polygons and defines the structure of the attribute table. This layer must be edited to define the polygons of different surface water hydraulic conveyance, if the polygons already exist in other layers, then copy and paste into this layer.

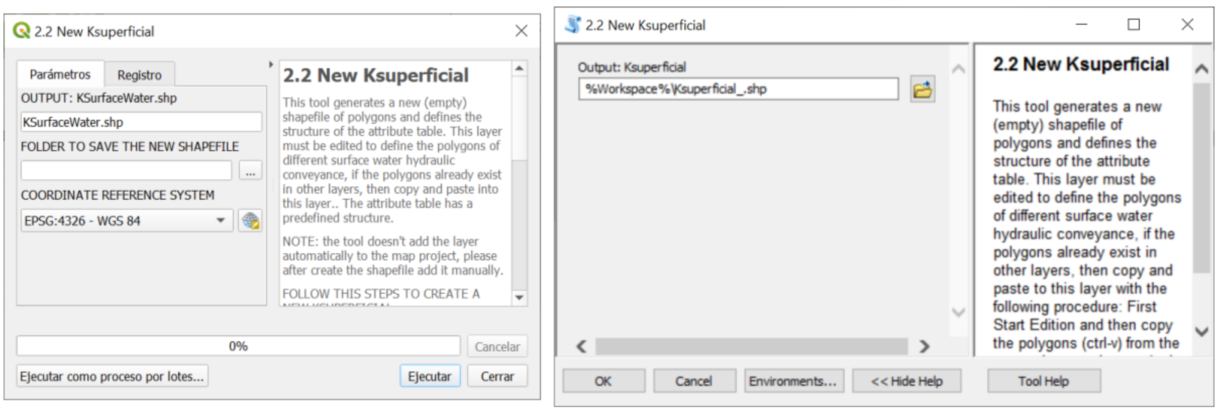


Figure 42: Tools to create a new empty layer Ksuperficial with the predefined attribute table structure in QGIS and ArcMap.

Usage of the tool (Figure 42)

FOLLOW THESE STEPS TO CREATE A NEW KSUPERFICIAL:

- Use this tool to create a new empty Ksuperficial layer. Save the layer in the project workspace created by the MELEF FSWmodel.
- Edit the layer to define zones with and without surface water to define different hydraulic conveyance (all nodes should be covered by polygons): you can copy polygons from other layers. ArcMap: First start edition and then copy the polygons (ctrl-v) from the source layer and paste (ctrl-c) to the target layer under edition.
- Fill with values the parameters in the attribute table.

ATTRIBUTE TABLE DESCRIPTION:

- INFO: field to fill with general purpose information about the polygon.
- KSUPERF: define a value between 1e-2 to 1e-7, where higher values defines more hydraulic conveyance (water bodies and rivers: 1e-2 to 1e-5; hillslopes with different degrees of vegetation: 1e-6 to 1e-8).

2.3.2.3 2.3 Generate SOIMELEF

Summary

This tool creates the shapefile SOIMELEF.shp with parameters of the Non Saturated Zones. The tool executes and Spatial Join operation between the layers *NonSaturatedZone*, *Ksuperficial* and *Nodes*. The attribute table of this layer is used by MELEF FSW Interface to generate the simulation file *.SOI.

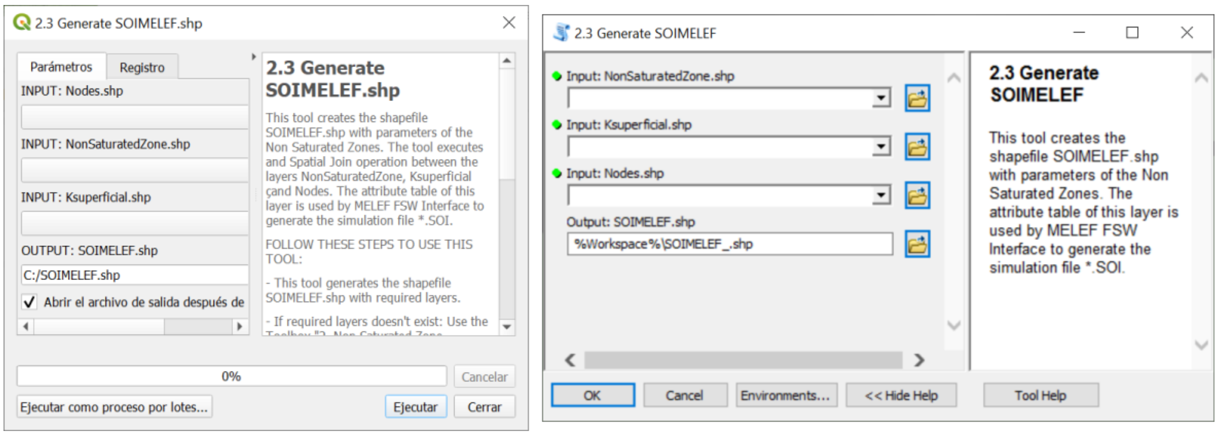


Figure 43: Tools to generate the simulation layer SOIMELEF in QGIS and ArcMap.

Usage of the tool (Figure 43)

FOLLOW THESE STEPS TO USE THIS TOOL:

- This tool generates the shapefile SOIMELEF.shp with required layers.
- If required layers doesn't exist: Use the Toolbox "2. Non Saturated Zone - EvapoTranspiration (SOI)" to create empty shapefiles with the required attribute table structure, and Start the Edition to define the polygons and information required.
- Select the layer NonSaturatedZone.shp with Soil Types and Land Cover properties, as well as the layer Ksuperficial.shp with the hydraulic conveyance of surface water.
- All nodes should have properties of both layers, if some layer doesn't intersect some nodes, then it creates an error that must be fixed before continuing.

2.3.3 3. Geological Materials - Impervious Substratum (PRN)

2.3.3.1 3.1 Approximate an Impervious Substratum

Summary

Use this tool to fill the SUSTRATO-field of the DTMsubstratum layer with the elevation of the user-defined impervious substrate.

This tool creates an Hypothetical Impervious Substratum (*HIS*) using the Ec.(1) to generate a gradient in function of max and min (Zmax, Zmin)elevations of the Digital Elevation Model and user defined aquifer thickness (MaxEsp and MinEsp). It is common to find, but it is not a rule, that in top hill/mountain areas the aquifer thickness is less than in foothill/piedmont or valleys/plan areas. If the user wants to use a constant aquifer thickness, then the values of MaxEsp and MinEsp should be the same to transform the Ec.(1) into Ec.(2).

Thus, the resultant *HIS* is the limit of the first material that start from the surface of the soil, and it is the material where the groundwater flows principally. Below the first material a minimum transmisivity is applied by the MELEF FSW model.

The hypothetical Impervious Substratum obtained from Ec(1) or (2) can be re-sampled to create a smoothing effect. Select one of available resampling methods to upscale the HIS raster and a cell size lager than the original DEM input raster. The larger the cell size, the the greater the smoothing effect. The other option is to use the interpolation methods proposed in the tool (available in ArcMap).

$$HIS = Z - (MaxEsp - (MaxEsp - MinEsp) \cdot (Z \cdot (Z - Zmin))/(Zmax \cdot (Zmax - Zmin)))Ec.(1)$$

Where in Ec(1):

HIS = Hypothetical Impervious Substratum elevation.
Z = Digital Elevation Model cell values.
MaxEsp = Aquifer thickness in foothill/piedmont or valleys/plan areas.
MinEsp = Aquifer thickness in top hill/mountain areas
Zmax = Maximum elevation in the DEM input raster.
Zmin = Minimum elevation in the DEM input raster.

If MaxEsp = MinEsp then the Ec.(1) transforms into:

ES = Z - MaxEsp Ec.(2)

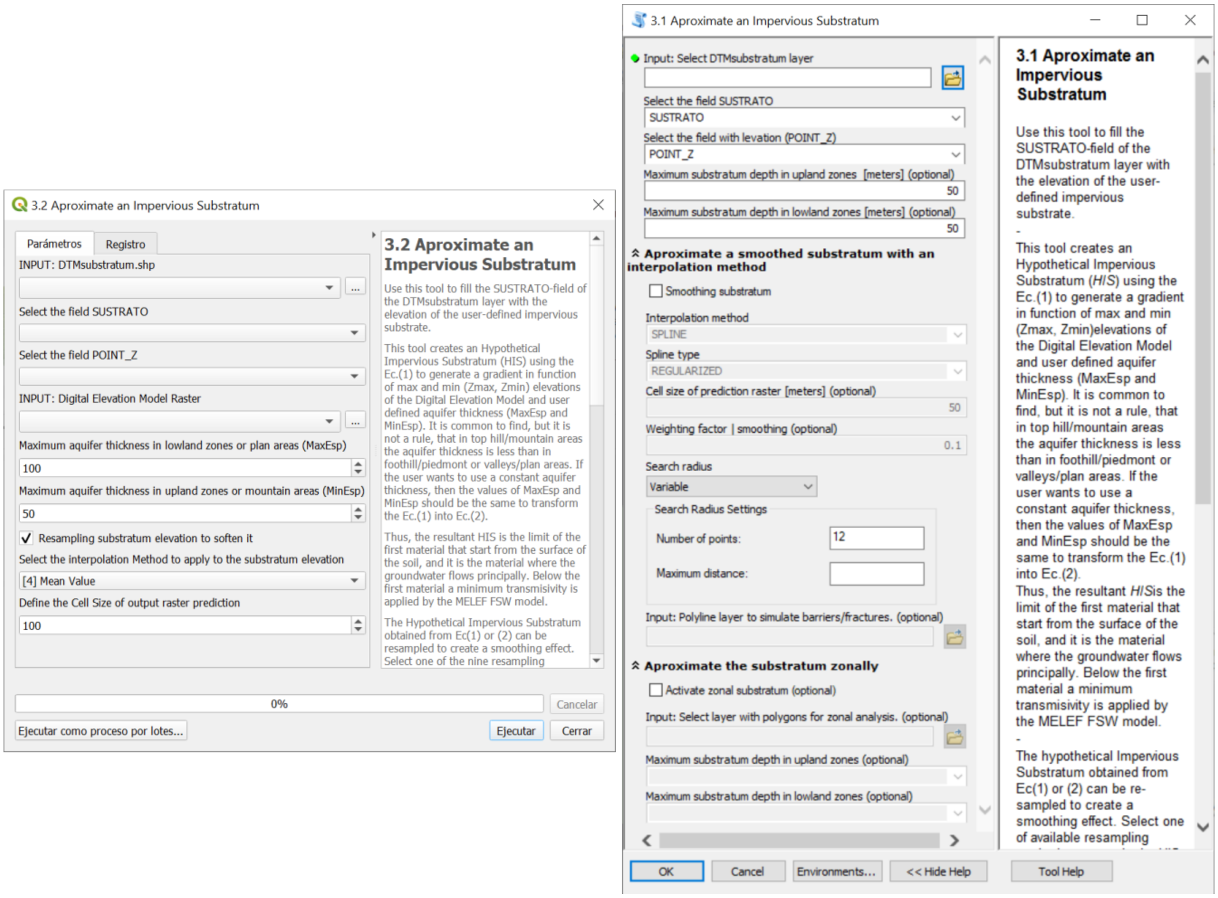


Figure 44: Tools to aproximate an impervious substratum in QGIS and ArcMap.

Usage of the tool (Figure 44)

FOLLOW THESE STEPS TO GENERATE AN APROXIMATION OF THE IMPERVIOUS SUBSTRATUM ELEVATION.

- Select the required layer DTMsubstratum to fill the SUSTRATO field with an approximation of the elevation of impervious substratum.
- The tool allows you to use a global or zonal evaluation of the impervious substratum
  - In case of global evaluation (QGIS-ArcMap): Define the maximum substratum depth in upland zones and the maximum depth in lowland zones of the catchment.
  - In case of zonal evaluation (ArcMap): create a layer of polygons (sub-basins) to make a zonal evaluation of the substrate, i.e., in each polygon you need to create and populate two fields with the maximum substratum depth in upland and lowland areas of the sub-basin.

- ArcMap: Optionally, you can active the option to smooth the output substratum elevation obtained from the zonal or global evaluation, the interpolation methods available are Spline; Spline With Barriers; IDW.
- QGIS: Select one of the nine resampling methods to upscale the substratum elevation raster and a cell size lager than the original input raster. The larger the cell size, the greater the smoothing effect.
- Important to check: the field WEATHERING is generated and populated with the difference between TERRAIN ELEVATION (COTA 1) and SUSTRATO, to confirm if all values are positive. In case there is negative value, then SUSTRATO elevation is higher than terrain elevation and you need to modify interpolation settings to avoid this condition.

2.3.3.2 3.2 New Geology

Summary

This tool generates a new (empty) shapefile of polygons and defines the structure of the attribute table. This layer must be edited to define the polygons of different geological materials, if the geology already exists in other layers, then copy and paste into this layer.

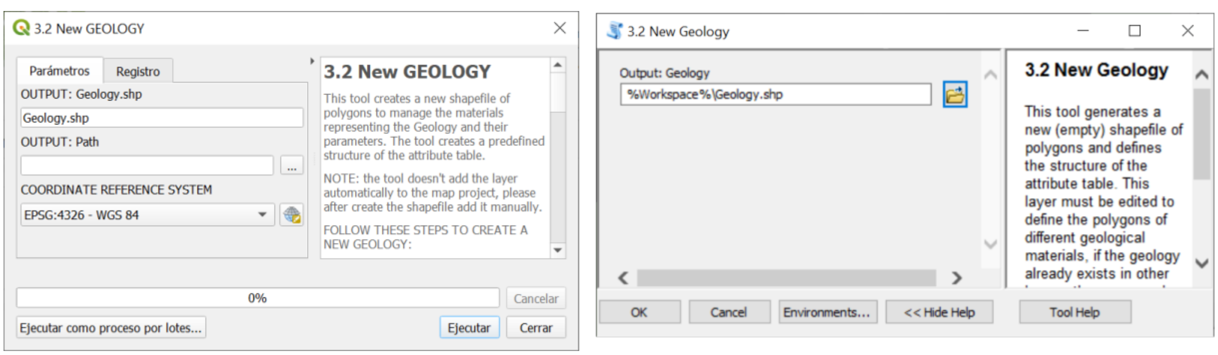


Figure 45: Tools to generate the new empty layer Geology with a predefined attribute table structure in QGIS and ArcMap

Usage of the tool (Figure 45)

FOLLOW THESE STEPS TO CREATE A NEW GEOLOGY:

- Use this tool to create a new empty Geology layer. Save the layer in the project workspace created by the MELEF FWS model.
- Edit the layer to define different geological materials: you can copy polygons from other layers.
- Fill with values the parameters in the attribute table.

ATTRIBUTE TABLE DESCRIPTION

- N: Effective porosity of materials.
- KX: Hydraulic conductivity of materials in the X axis in units of m/second if it is not defined in the field UNIDADES.
- KY: Hydraulic conductivity of materials in the Y axis in units of m/second if it is not defined in the field UNIDADES.
- KZ: Hydraulic conductivity of the soil in the Z axis, It could be considered as the infiltration capacity, in units of m/second if it is not defined in the field UNIDADES.
- ALFA: Angle of anisotropy of the hydraulic conductivity in degrees. It controls the direction of X and Y axes measured from north and counterclockwise.
- ZONA: Polygon identification zone.
- UNIDADES: Units of hydraulic conductivity parameters, keep this field empty to use the default units of meters per second (m/s) or define other units like meters per day (m/d).
- TIME_FILE: Define a column name of an Excel file with transient values of any parameter of this layer.

2.3.3.3 3.3 Generate PRNMELEF

Summary

This tool creates the shapefile PRNMELEF.shp with parameters if geological materials and location of the impervious substratum. The tool executes a Spatial Join operation between the Layers MDTSustrato and Geology.The attribute table of this layer is used by MELEF FSW Interface to generate the simulation file *.PRN.

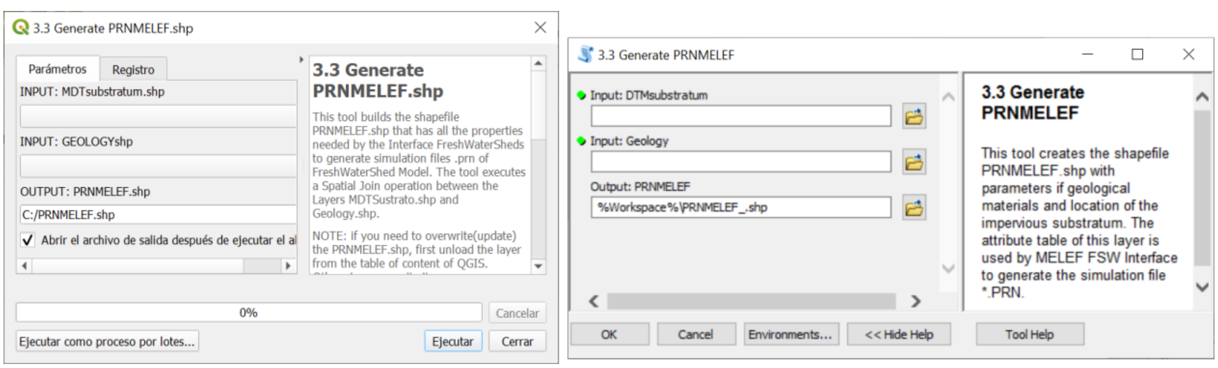


Figure 46: Tools to generate the simulation layer PRNMELEF in QGIS and ArcMap.

Usage of the tool (Figure 46)

FOLLOW THESE STEPS TO CREATE THE PRNMELEF

- This tool generates the shapefile PRNMELEF.shp with required layers.

- If required layers doesn't exist: Use the Toolbox "3. Geological Material - Impervious Substratum (PRN)" to create empty shapefiles with the required attribute table structure and Start the Edition to define the polygons and information required.
- Select the layer Geology.shp with geological material properties and DTMsubstratum with information about terrain elevation and elevation of the impervious substratum.
- All nodes should have properties of both layers, if some layer doesn't intersect some nodes, then it creates an error that must be fixed before continuing.

2.3.4 4. Boundary and Inner Conditions (CND - SLC)

2.3.4.1 4.1 New CND - SLC

Summary

This tool creates an empty CND.shp or SLC.shp polygon layers with a predefined attribute table structure.

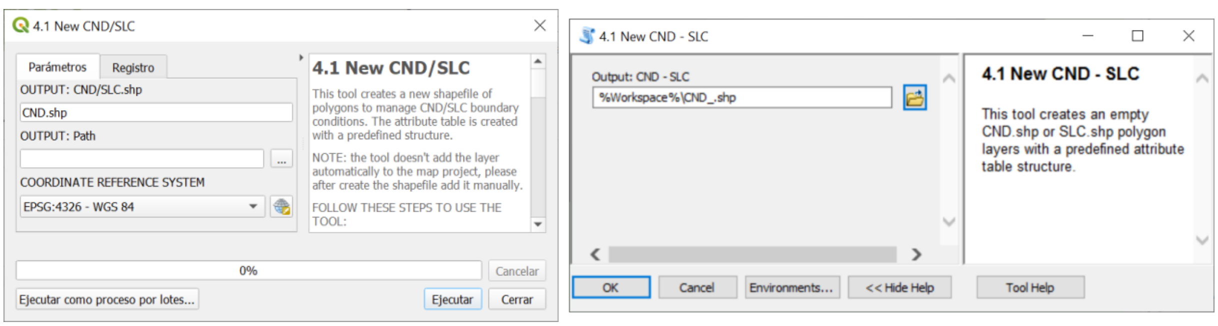


Figure 47: Tools to create a new empty layer CND or SLC with the attribute table structure predefined in QGIS and ArcMap.

Usage of the tool (Figure 47)

FOLLOW THESE STEPS TO USE THIS TOOL:

- It is recommended to use the suggested names to avoid confusion in the following tools that ask for these layers as input files.
- ArcMap: Define the Workspace variable in Geoprocessing>Environments...>Workspace
- Edit the CND or SLC layer to draw polygons over the nodes with a boundary condition.
- If the SLC layer is not required to establish a punctual lateral injection on the mesh, then do not generate this layer.

ABOUT THE LAYERS:

- CND: Create a CND layer if the required boundary condition is to impose a continental boundary water level (surface or ground), a continental-coastal water-level (surface or ground), boundary diffuse lateral in/outflow or impose a boundary water table gradient.
- SLC: Create a SLC layer if the required boundary condition is to impose a boundary punctual lateral injection of water to the aquifer.

ATTRIBUTE TABLE DESCRIPTION

- INFO: Describe in this field the boundary condition defined.
- CODIGO_USO: Select the right code number to define the boundary condition type to impose:
  - 1 (CND): Imposed fresh water level on the mesh boundary [meters]
  - 11 (CND): Imposed
  - 2 (CND): Imposed diffuse lateral flow on the mesh boundary [m3/m_lineal/sec]
  - 3 (CND): Imposed water table gradient on the mesh boundary, define negative values for downstream direction [m/m].
  - 7 (SLC): Punctual lateral injection of water to the aquifer [m3/sec].
- VAL_PERM: Permanent value of the imposed condition
- TIME_FILE: Define the name of the column in an Excel file with the transient behavior of boundary or inner condition of this layer.
- ZONA: Polygon identification zone.
- UNIDADES: Define the units of the permanent value of the field VAL_PERM if it is different than the default units. Maintain this field empty to use default units.

2.3.4.2 4.2 Generate CNDMELEF or SLCMELEF

Summary

This tool executes and spatial join operation between layers CND/SLC and Nodes to generate the layers CNDMELEF or SLCMELEF required by MELEF FSW model to generate Boundary/Inner Conditions simulation file *.CND or *.SLC.

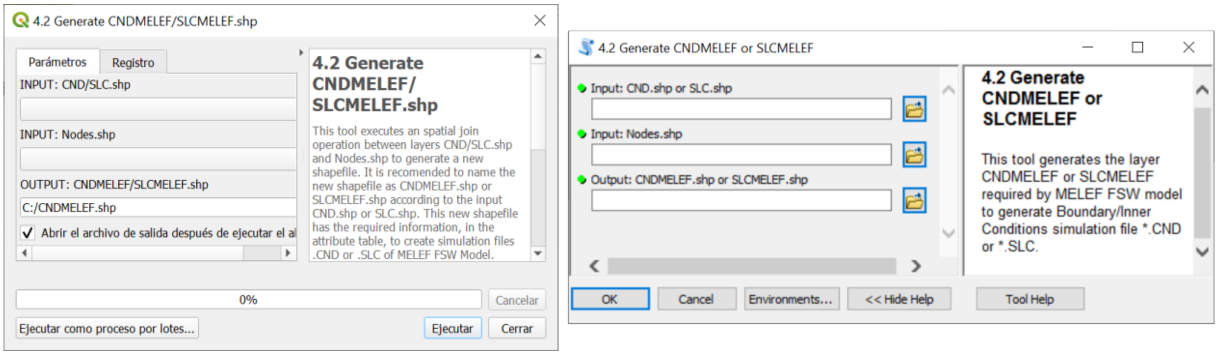


Figure 48: Tools to generate the simulation layers CNDMELEF and SLCMELEF in QGIS and ArcMap

Usage of the tool (Figure 48)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Define the required input files.
- Define the path and name of the output shapefile CNDMELEF or SLCMELEF in function of the inputted CND or SLC shapefile.

2.3.5 5. Rainfall and Water Uses (SLR)

2.3.5.1 5.1 New Rainfall - Water Uses

Summary

This tool creates an empty Rainfall or Water Uses polygon layers with a predefined attribute table structure.

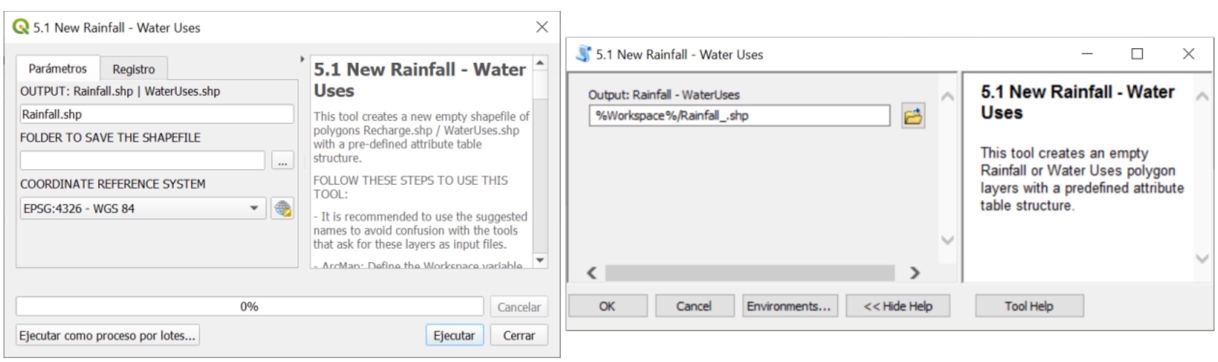


Figure 49: Tools to generate the new empty layer Rainfall/WaterUses with the attribute table structure predefined in QGIS and ArcMap.

Usage of the tool (Figure 49)

FOLLOW THESE STEPS TO USE THIS TOOL:

- It is recommended to use the suggested names to avoid confusion with the tools that ask for these layers as input files.
- ArcMap: Define the Workspace variable path in Geoprocessing>Environments...>Workspace.
- Edit the Rainfall or Water Uses layer to draw polygons over the nodes with these- conditions. -Example 1: Create a Thiessen polygon layer with the point layer location of climatic stations and create the layer Rainfall with these polygons. -Example 2: Create the layer Water Uses and start edition to draw multiple polygons over the nodes where there is a pumping well, injection well, surface water diversion, or other water use.

ABOUT THE LAYERS:

- Rainfall: Use this layer to define precipitation or irrigation in the analysis area. This layer should cover with polygons all the nodes to avoid an error in the MELEF FSW model.
- Water Uses: Use this layer to define water uses with polygons only over the nodes with a simulation condition.

ATTRIBUTE TABLE DESCRIPTION

- INFO: general description of the water uses, or rainfall gauging station defined.
- CODIGO_USO: code of use that defines the simulation condition imposed, rainfall or water use as follows (see the manual for more details):
  - 4 (Water Use): Select this code of use value to simulate punctual pumping well (+ positive value for extraction of groundwater) or punctual injection well (- negative value for injection of water to the aquifer) [m3/sec]. This condition assumes that groundwater can only be extracted from the saturated aquifer thickness, meaning that it is not possible to extract water below the elevation of the impervious substratum or extract water if the freshwater elevation is the same than the sea water wedge in coastal modeling.
  - 5 (Rainfall): Select this code of use value to simulate rainfall or diffuse irrigation (- define negative values) [m/sec].
  - 6 (Water Use): Select this code of use to simulate surface water diversion, use + positive value to define the rate of diversion with units of m3/m2/day. IMPORTANT: the total area of influence of the nodes should be considered in the evaluation of surface water diversion rate.
  - 8 (Water Use): Select this code of use to simulate a water drainage circular pipe. Consider the following to impose this simulation condition, fill the field VAL_PERM with the permanent diameter of the pipe in units of centimeters and consider that the pipe is simulated without slope and that the outflow is due to the hydraulic load of surface water only. In case the pipe has a gate,

then use the TIME_FILE to define in an Excel a column with the transient opening of this gate to reduce the diameter to the gate opening.
  - 9 (Water Use): Select this code of use to simulate spillway/sluice gate in a reservoir or other hydraulic structure for surface flow control. Consider the following to impose this simulation condition, fill the field VAL_PERM with the permanent value of the gate opening in meters, or use the field TIME_FILE to define a transient opening of the gate.
- VAL_PERM: use this field to define a permanent value of Rainfall or Water Use condition.
- TIME_FILE: use this field to define a column name in an Excel file with transient behavior of Rainfall or Water Use condition.
- ZONA: Polygon identification zone that is automatically managed.
- UNIDADES: use this field to define distinct units than default units. The default units are described in each code of use.
- ZONAINYEC: use this field to define a bypass injection zone of water. To enable the bypass first to define a Water Use with a code of use [6 or 8] and then create a new polygon where the ZONAINYEC should have the identification value of the field ZONA with the Water Use condition. The result is that the surface water diverted or drained by a pipe is bypassed to the required nodes.

2.3.5.2 5.2 Generate SLRMELEF

Summary

This tool generates the layer SLRMELEF required by MELEF FSW model to generate simulation conditions*.SLR.

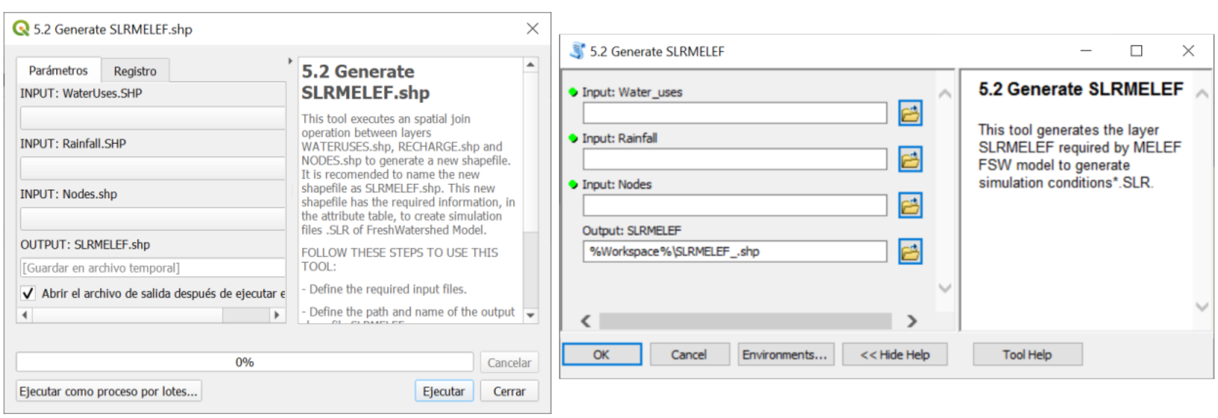


Figure 50: Tools to generate the simulation layer SLRMELEF in QGIS and ArcMap.

Usage of the tool (Figure 50)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Define the required input files.
- Define the path and name of the output shapefile SLRMELEF.

2.3.6 6. Gauging Sections (SEC)

2.3.6.1 6.1 New Gauge Section

Summary

This tool creates a new Gauge Section polyline to define the gauge sections of the Project.

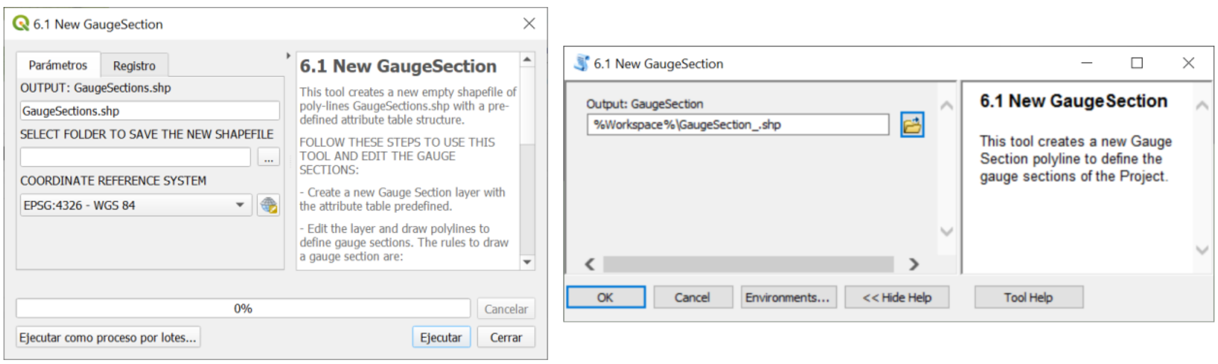


Figure 51: Tools to create the new empty layer GaugeSection with the attribute table structure predefined in QGIS and ArcMap.

Usage of the tool (Figure 51)

FOLLOW THESE STEPS TO USE THIS TOOL AND EDIT THE GAUGE SECTIONS:

- Create a new Gauge Section layer with the attribute table predefined.
- Edit the layer and draw polylines to define gauge sections. The rules to draw a gauge section are:
  - Follow the borders of the triangular elements as the path to draw the polyline gauging sections.
  - Draw a polyline vertex every time it crosses a node.
  - Draw the gauging section from the right bank and towards the left bank looking downstream of the surface flow. In this way the flow will be positive, and negative in the opposite direction.
  - Draw gauge sections enclosing water bodies or areas of interest to perform a water budget (finish the last vertex of the gauging sections on the first vertex drawn).
  - Gauge section results of MELEF FSW model are calculated only during the execution of the model.

ATTRIBUTE TABLE DESCRIPTION:

- INFO: fill this field with information about the gauge section.
- NNUDO: number of nodes that is automatically managed.
- VAL_PERM: this field is not in use, maintain empty this field.
- ZONA: this field is the identification zone value of each gauge section; it is automatically managed.

2.3.6.2 6.2 Generate SECMELEF

Summary

This tool generates the SECMELEF layer required by MELEF FSW to generate the simulation file *.SEC.

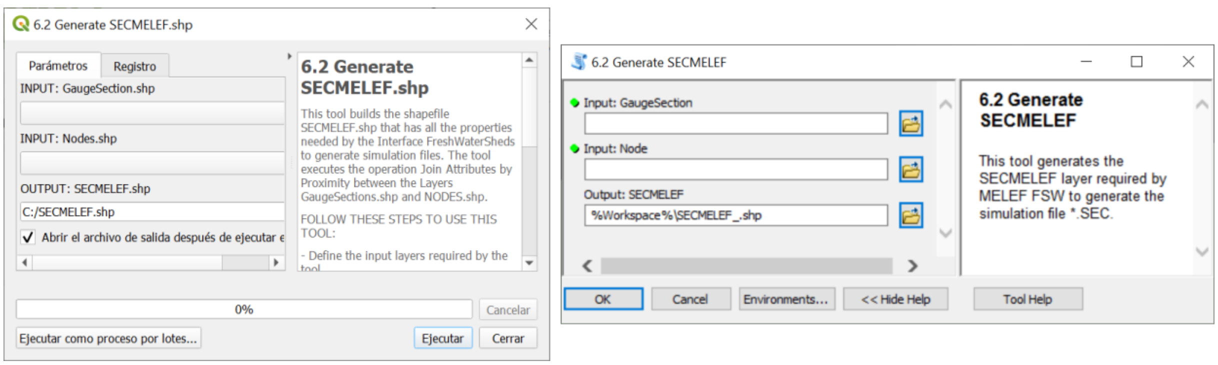


Figure 52: Tools to generate the simulatin layer SECMELEF in QGIS and ArcMap.

Usage of the tool (Figure 52)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Define the input layers required by the tool.
- Check if all gauging sections required are drawn in GaugeSection layer before run the MELEF FSW model.
- ArcMap: Define the Workspace variable of the Project. Geoprocessing>Environments...>Workspace>Current Workspace | Scratch Workspace
- Define the namefile of the output layer. It is suggested to use the predefined names.
- Execute the tool to create the SECMELEF layer. In the MELEF FSW Interface use the *.dbf file associated to this layer to create the simulation gauging sections file *.SEC.

2.3.7 Utilities (ArcMAP ESRI)

2.3.7.1 Adjust Infiltration Rate

2.3.7.1.1 Adjust KZ by Slope

Summary

Use this tool to adjust the KZ field in function of the slope. This tool uses the cosine function to reduce the infiltration capacity when the slope increases and reduce when the slope decrease. The equation solved is KZ = (KZ * Cos(slope)) / Avg(Cos(Slope)), and the original average infiltration capacity value is maintained after adjust it in function of the slope.

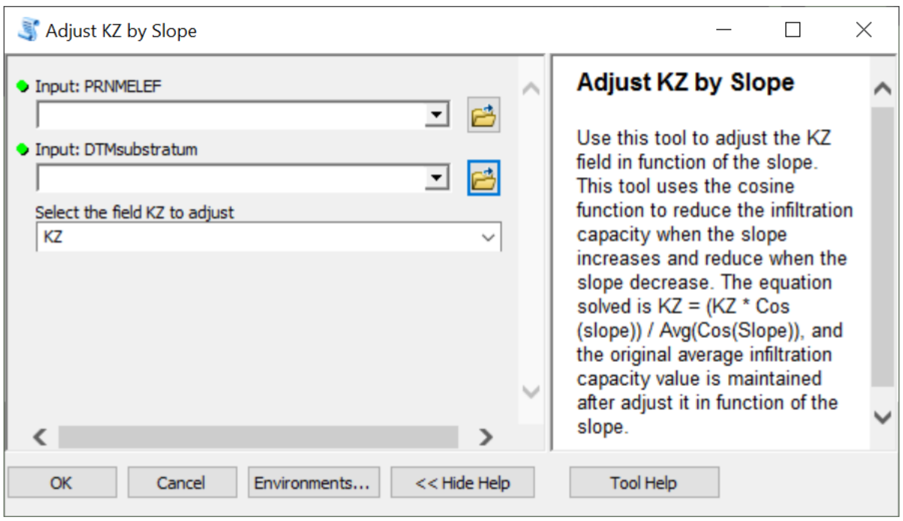


Figure 53: Tool to adjust the maximum infiltration capacity of the soil (Kz) in ArcMap.

Usage of the tool (Figure 53)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Select the required layers PRNMELEF and DTMsubstratum
- Select the field KZ whit the infiltration capacity values to adjust in function of the slope.
- The layer PRNEMELF must have the fields AREA_M2 and SLOPE.

2.3.7.2 Adjust Soil Thickness

2.3.7.2.1 Adjust ES by Slope and Vegetation

Summary

Use this tool to adjust the Soil Thickness (ES, for its Spanish acronym) in function of the maximum hillslope length, maximum and minimum Soil Thickness and Vegetation index. The tool can perform a global or zonal evaluation. The illustration Figure 54 shows the ES adjusted for different values of vegetation factor Fveg.

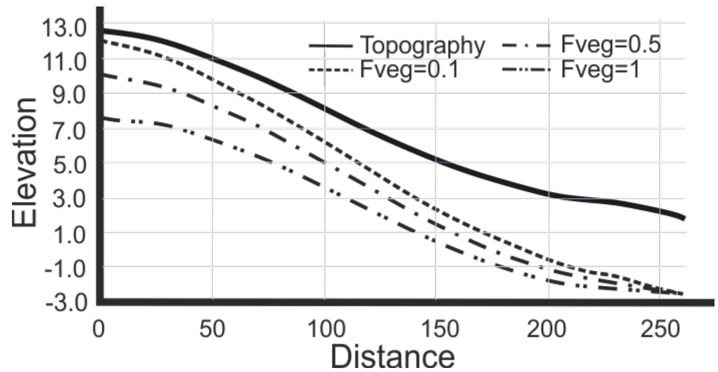


Figure 54: Evolution of the adjusted Soil Thickness using different parameter configuration of the vegetation factor Fveg.

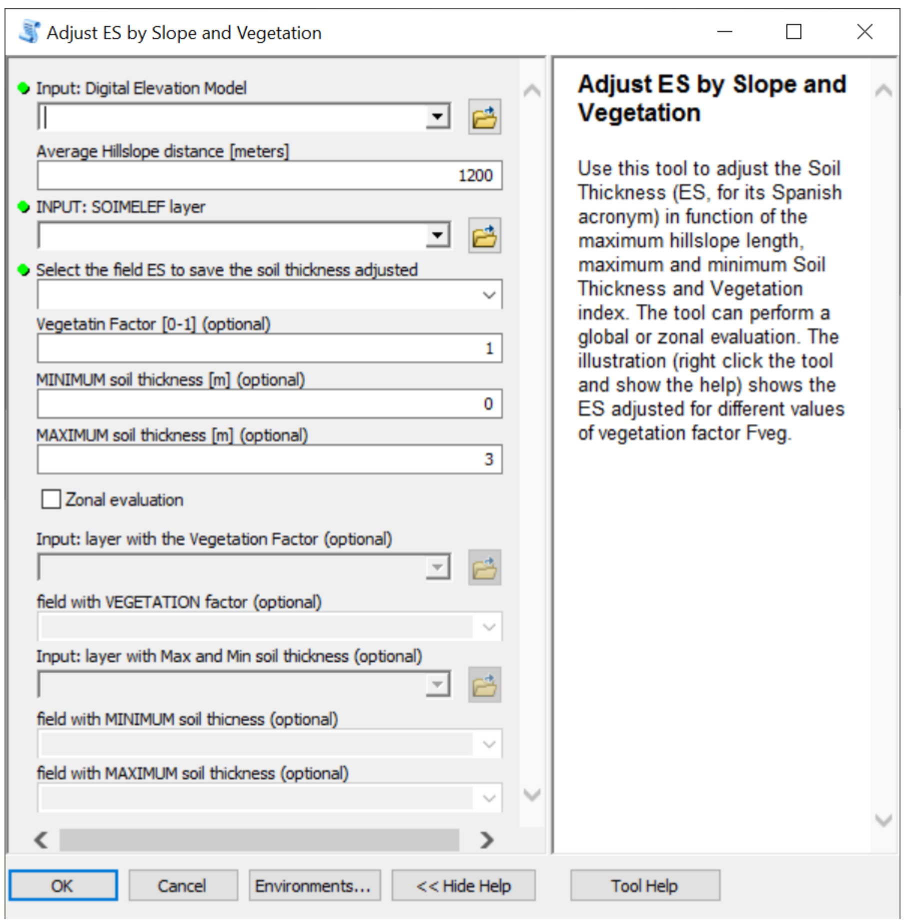


Figure 55: Tool to adjust the Soil Thickness in function of Slope and a vegetation factor.

Usage of the tool (Figure 55)

Use this tool when there is not observed information about the Soil Thickness distribution in the study area. The tool approximates the Soil Thickness with the following equation:

ES = ESmin + [(ESmax - ESmin) cos(Phi * Pi / 180) (Fdist + (1 - Fdist) Fveg)]

Where:

- ES = Soil thickness (ES, by its Spanish acronym)
- ESmin = Minimum soil thickness expected along the hillslope profile.
- ESmax = Maximum soil thickness expected along the hillslope profile.
- Phi = Slope of the hillslope profile in degrees.
- Fdist = Distance factor, the closer one is to the beginning of the hillslope it will tend to zero, while at the foot of the slope this value will tend to 1
- Fveg = Vegetation factor with values between 0 -1. Values close to 1: soil thickness is close to the maximum value. Values close to 0: soil thickness is close to the minimum value.

FOLLOW THESE STEPS TO USE THIS TOOL:

- If it is a global adjustment of soil thickness: then define global values for maximum and minimum expected soil thickness (ES), vegetation factor (Fveg) and the average distance factor (the average distance between top hillslope and foot hillslope).
- If it is a zonal adjustment of soil thickness:
  - Create a polygon layer and define the values of vegetation factor [0-1] (create a field in the attribute table with the Fveg values). This polygon layer could be the vegetation type land cover, where maximum depth of roots could be used to define the vegetation factor between vegetation types.
  - Create a polygon layer and define the values of Maximum and Minimum expected soil thickness (create two fields in the attribute table with the ESmin and ESmax). For example, this polygon layer could be defined with sub-basins and areas with surface water bodies where soil thickness can accumulate.

2.3.7.3 Generate Simulation Files

2.3.7.3.1 Generate simulation file COR

Summary

Use this tool to generate the simulation file *.COR using a point layer with the coordinates of the mesh nodes.

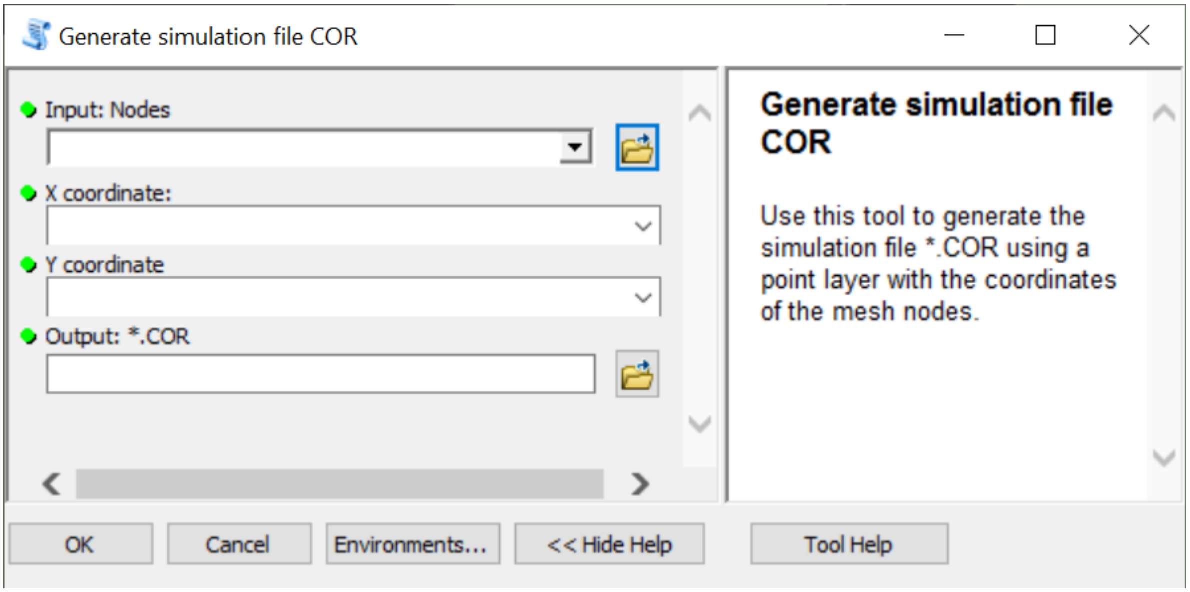


Figure 56: Tool to create the simulation file COR in ArcMap.

Usage of the tool (Figure 56)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Select the layer NODES of the mesh.
- ArcMap: If the point layer doesn't have the coordinates of the nodes in the attribute table, then apply the tool AddXY Coordinates to add the coordinates.
- Select the fields with the corresponding coordinate X and Y.
- Define the output file path and name.

2.3.7.3.2 Generate simulation file INI

Summary

Use this tool to generate a new *.INI file with the initial solution of water-table level in each node. The MELEF FSW model requires an initial solution to start the simulation.

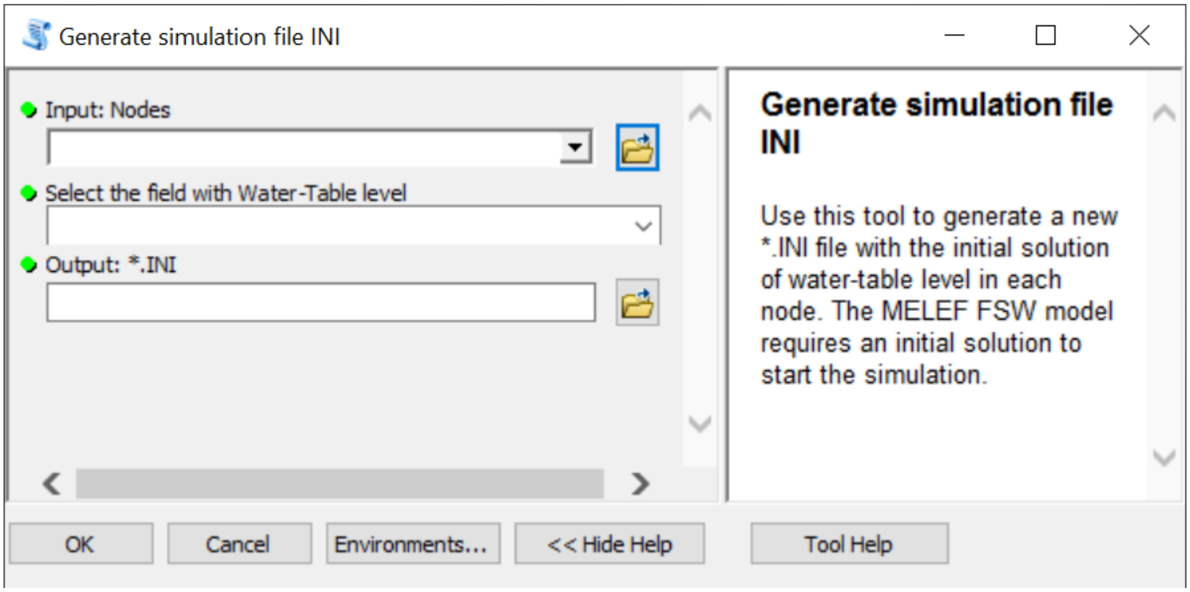


Figure 57: Tool to generate the simulation file .INI with the initial solution for the model MELEF FSW.

Usage of the tool (Figure 57)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Select a shapefile of points with the nodes of the mesh.
- Select from the attribute table of the shapefile selected the field with the water-table elevation values.
- Define the output path and name file with the extension *.INI.

NOTE: There are different possible ways to generate the initial solution, one of these possibilities is to use groundwater levels measured in a well monitoring network and use an interpolation method to create an interpolation raster. Use this interpolation raster with the tool to create a 3D shapefile and use a tool to ADD XY COORDINATES (it adds the coordinates including the elevation Z). Use this 3D shapefile with the Z coordinate to create the file .INI.

2.3.7.4 Reconditioning DEM and MESH

2.3.7.4.1 DEM Burning by Slope

Summary

This tool can be used with polylines representing streams or sections of streams to fix often DEMs errors caused by structures that obscure the topography of the original terrain, such as bridge decks, which are reflected in the DEM as obstructions to water flow. Using a hydrologically incorrect DEM leads to an incorrect simulation of the hydrology of the site, so it is necessary- to recondition the areas through which the river channel should pass- and avoid false dams or lakes.

The tool drops- the cell elevation of a DEM by slope, it is done following the trace of a polyline layer. The polyline layer- vertices- are used to save the Digital Elevation Model (DEM), and the distance between vertices is resampled- to the half cellsize of the DEM. The tool iterates for each- polyline feature part and modifies the vertices elevation to maintain a downward slope following the start and end direction of each- polyline. If the slope between the start and end vertices is ascending, then an iteration starts- to dismiss the end vertices that have a higher elevation than the initial vertex and evaluate with the rest of vertices the average slope of the polyline feature- part. The average slope of the polyline feature- part- is used to modify the elevation of in between vertices with ascending slope see Figure 58. If the entire polyline feature part has an average ascending slope, then an error is shown with the following possible cause: the polyline was drawn following a direction opposite to the water flow direction in a slope. Finally, the elevation of the polyline vertices are converted to raster, and the cells of the DEM that intersect them- are updated with the new elevation burned by slope.

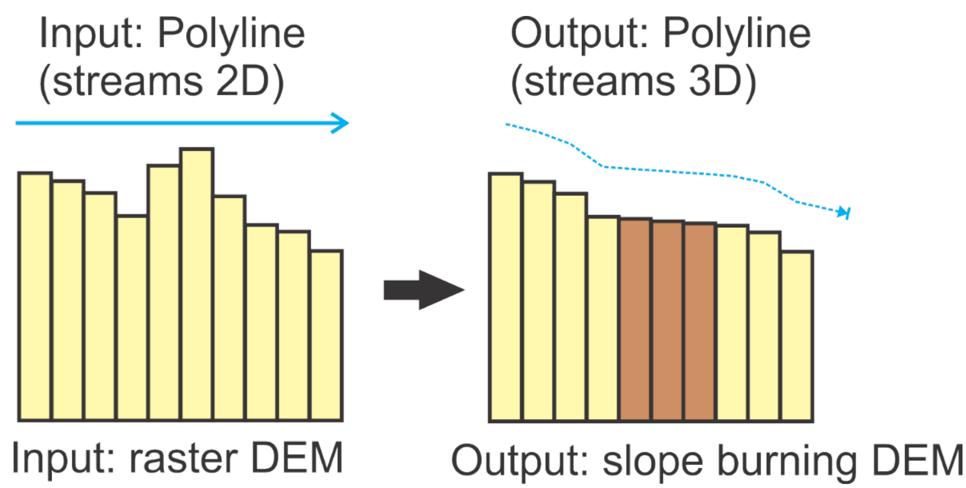


Figure 58: Representation of the Dem Burning Process by Slope over a stream profile.

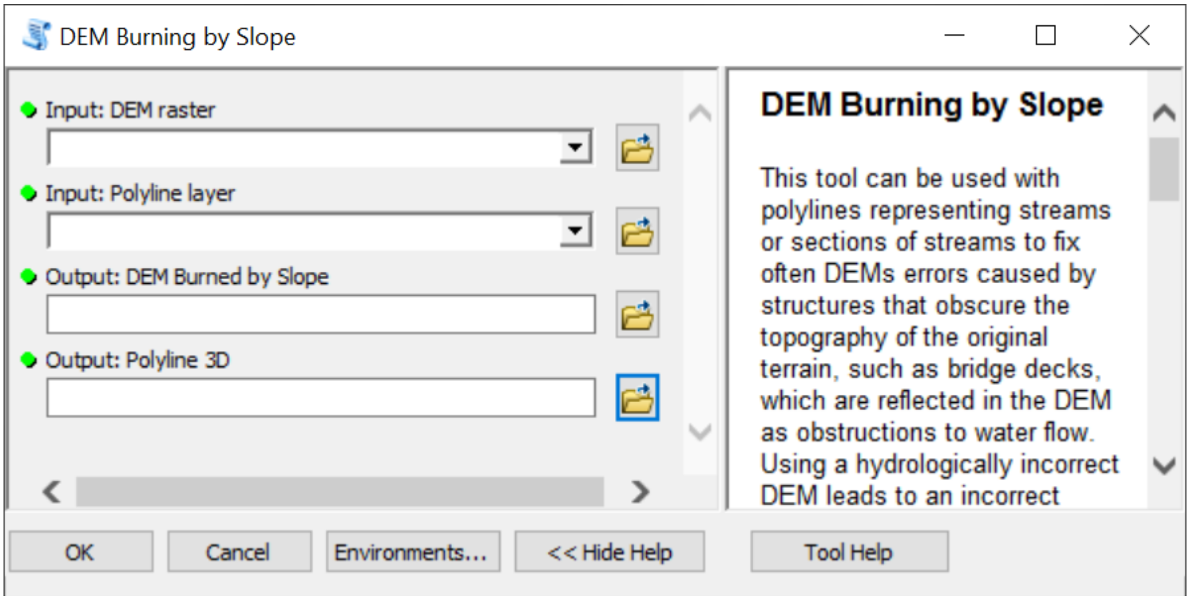


Figure 59: Tool to run the DEM Burning by Slope process and recondition a DEM raster to create hydrologically correct DEM.

Usage of the tool (Figure 59)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Create a new polyline layer with the following options:
  - Edit the polyline layer and draw polylines, from higher to lower elevation. This requires a manual detection of stream sections where the DEM has structures that obscure the topography of the original terrain and generate and ascending slopes in the downstream direction.
  - Use GIS well known process to define the polyline streams of the study area and use it to check if the DEM requires to burn the elevation by slope.
- Select a Digital Elevation Model as a source of elevation information.
- Define the output polyline3D path and namefile. This polyline3D has in its vertices the final burned elevation by slope, and this layer can be implemented in other tools like DEM RECONDITIONING MULTIPLE to redefine with the AGREE DEM METHOD the section shape of the streams.
- Define the output DEM path and namefile.

2.3.7.4.2 DEM Burning Elevation

Summary

Use this tool to DROP/RAISE the cell elevation of- a Digital Elevation Model (DEM) with a constant value defined in a polyline section. The tool permits to define by polyline sections the width and the drop/raise value. The result is a DEM burned with a polyline- layer of different width and drop/raise values- defined by user.

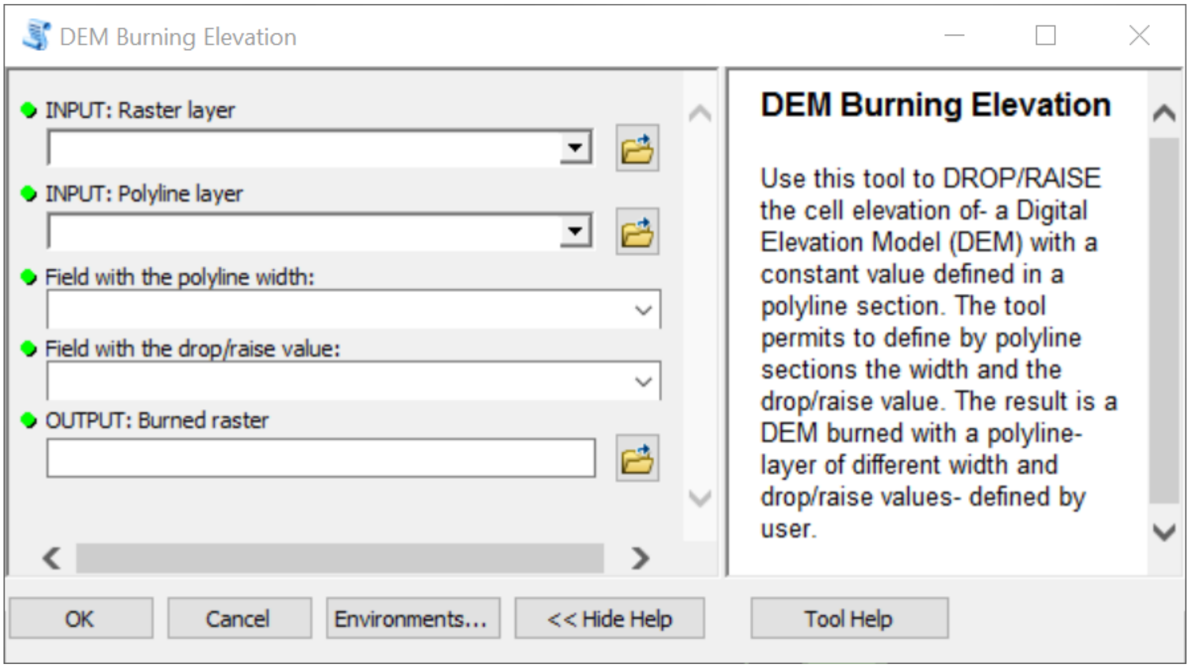


Figure 60: Tool to execute the process of DEM Burning Elevation for recondition of a DEM raster and create a hydrologically correct raster.

Usage of the tool (Figure 60)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Prepare a polyline layer and define the width and the drop/raise- value required for the DEM burning process. NOTE: Include this parameter in the attribute table of the polyline layer- and consider a positive value for the width variable and -/+ to define the drop/raise- parameter value- respectively.
- Select the raster layer that requires the DEM Burning process.
- Select the field of the polyline layer that has the width of the polyline (this distance is divided by two before generate the left and right buffers).
- Select the field with the value to drop/raise in the cells of the raster file that intersect the polyline.
- Define the output path and name of the burned raster.

2.3.7.4.3 DEM Reconditioning Multiple

Summary

This tool executes the AGREE DEM Method using a polyline layer to define their parameters in each polyline part. The polyline layer represents the streams network and the cells of the DEM below and close to the streams are modified with the

AGREE DEM method for stream reconditioning (change the channel cross section shape as it is represented in Figure 61). This tool differs from the original DEM Reconditioning_archydro version in that it allows defining the parameters of the AGREE DEM method from the attribute table of the polyline layer, it implies that different parameter settings can be defined for each polyline part.

DEM: Digital Elevation Model

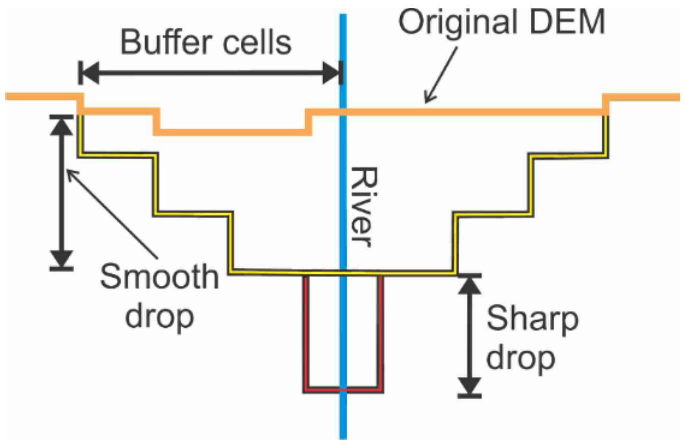


Figure 61: Representation of the Agree Method variables and influence over a stream section.

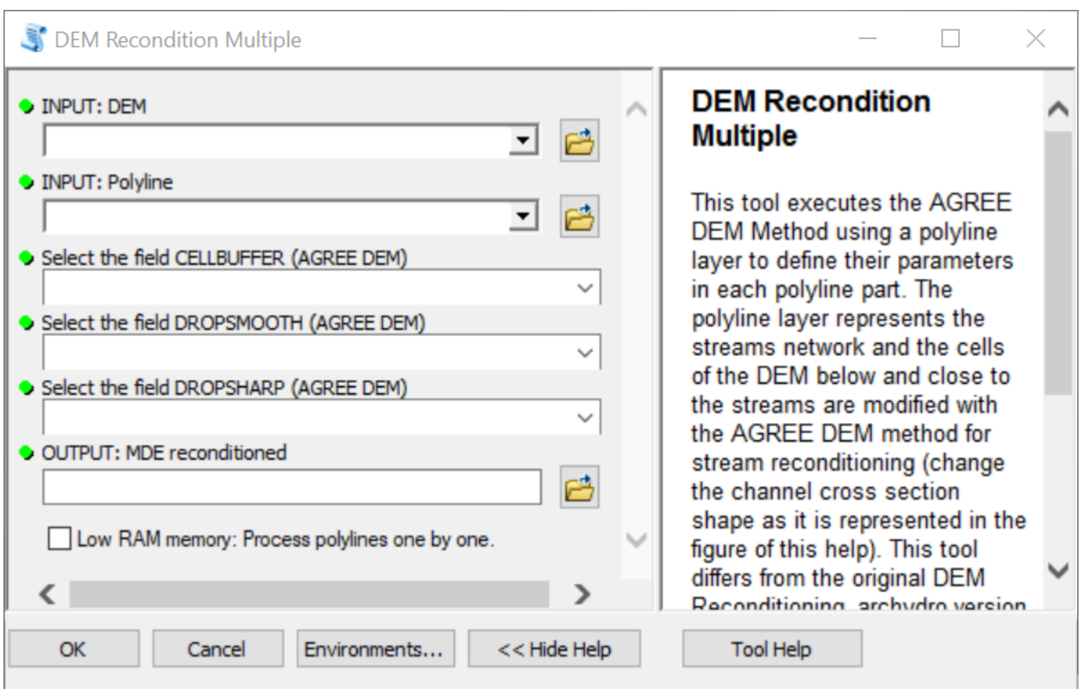


Figure 62: Tool to execute the DEM Recondition Multiple process and re-define the riverbed sections perpendicular to the streams path.

Usage of the tool (Figure 62)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Select a DEM with a cellsize representative of the average width of the streams that require reconditioning. Larger cell sizes imply less resolution, and the reconditioning process can create important modifications in stream flood plains.
- Select a Polyline layer that represents the streams that require the AGREE DEM reconditioning process. It is recommended to use a polyline layer with a tracing verified by orthophotos or other reference spatial information.
- Define the parameters of the AGREE DEM Method in the attribute table of the polyline layer, the meaning of each parameter is as follows:
  - CELLBUFFER: This parameter defines the number of raster cells that requires reconditioning from the stream centerline to the left and right sides of the stream. Note that the total width to be reconditioned is this parameter multiplied by two and by the size of the cells.
  - DROPSMOOTH: This parameter defines the smooth drop of the cell elevation value in the influence zone of the CELLBUFFER parameter.
  - DROPSHARP: This parameter defines the sharp drop of the cell elevation value that is just below the polyline trace. Note that this parameter has effect once the DROPSMOOTH is applied, it means that the total drop of the cell elevation just below the polyline trace should be the sum of DROPSMOOTH + DROPSHARP.
- Define output path and namefile of the raster DEM that results from the AGREE DEM Reconditioning process.

2.3.7.4.4 Fill Sinks in DEM

Summary

Use this tool to fill sinks of the Digital Elevation Model (DEM) in raster format. the tool requires a polygon layer with the topographic depressions where the field IsSink defines which areas should be filled and which ones not.

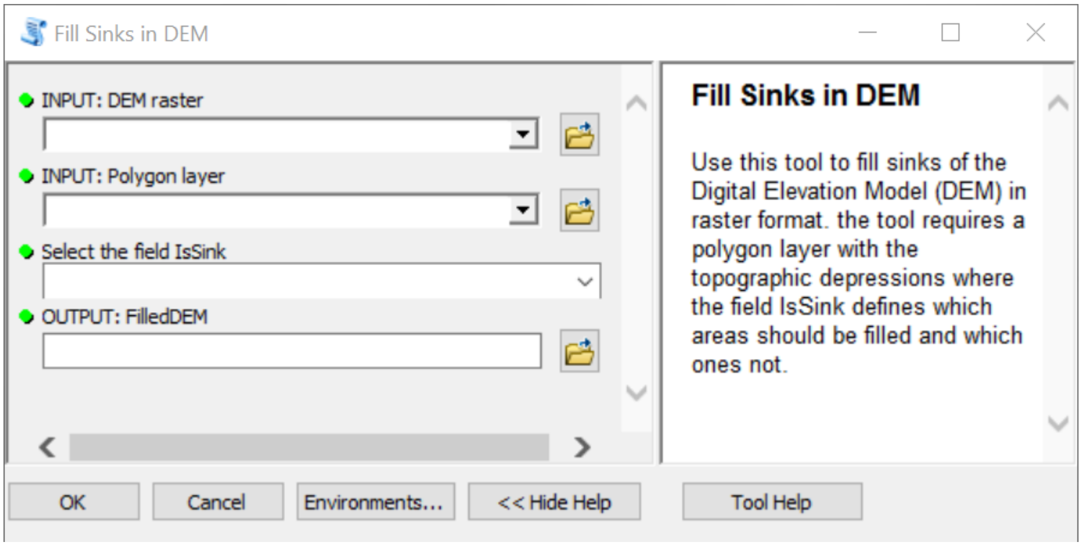


Figure 63: Tool to fill sinks/depressions in the Digital Elevation Model.

Usage of the tool (Figure 63)

FOLLOW THESE STEPS TO USE THIS TOOL:

- First run the tool "Topographic Depressions" and define in the attribute table of this layer which polygons are real sinks (IsSink = 1) and which polygons are false sinks (IsSink = 0). The tool will fill all the polygons where the field IsSink = 0.
- Run this tool using the output layer with the topographic depression polygons.
- Select the field IsSink of the polygon layer selected.
- Define the output path and namefile of the filled DEM.

2.3.7.4.5 Generate Buffers to Create Mesh Zones

Summary

Use this tool to generate multiple buffers and create zones where the finite element mesh must change the density of nodes. The primary source layer is a polyline layer with geological structures like faults, and hydrological elements like streams, dams or natural water bodies, The tool has three parameters, MEANWIDTH of the streams, MAXDIST of nodes required in the mesh and ALPHA that defines how fast the density of nodes decreases until reach the MAXDIST value.

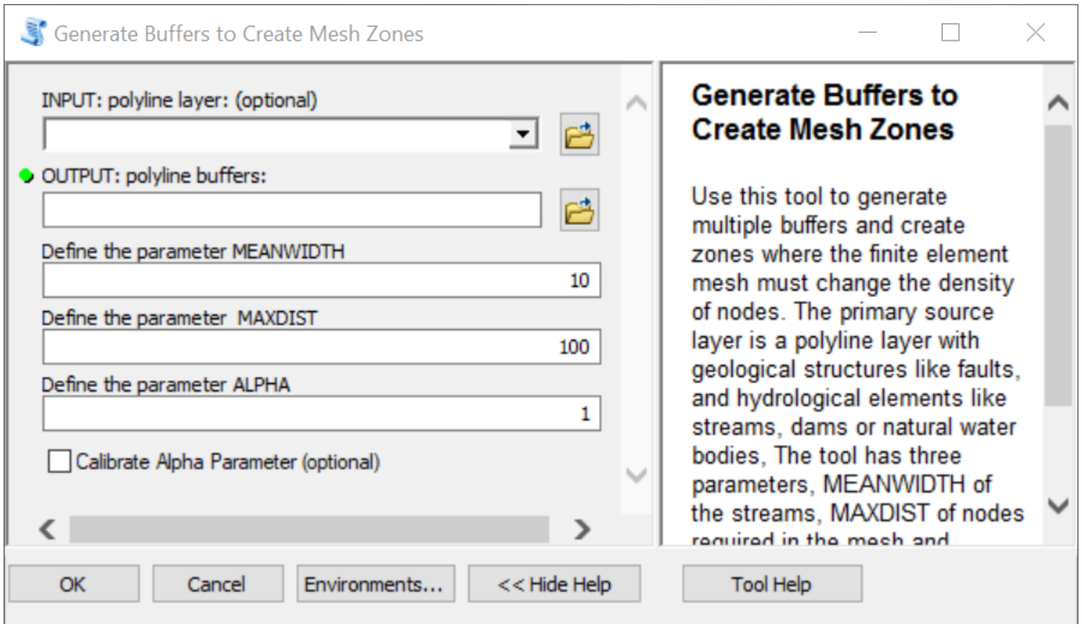


Figure 64: Tool to generate buffers along streams and control the transition between areas with high to low density of mesh nodes.

Usage of the tool (Figure 64)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Calibrate first the parameter configuration of MEANWIDTH, MAXDIST and ALPHA. To do this check the option "Calibrate Alpha Parameter" and run different configuration of parameters. The tool prints in the message window the number of buffers required for the parameter configuration selected. NOTE that ALPHA values can be between 0 and 2, where values close to zero has a gradual increment and values close to 2 has a rapid exponential evolution.
- Once the parameter configuration was calibrated, select the polyline layer with the geological structures and hydrological elements where is required to increase the nodes density and run again this tool without check the box *Calibrate Alpha Parameter*.
- Define the output path and namefile of the buffers created by this tool. Edit this buffers to clip them to the catchment limits and close them to form closed polygons if use the Aquaveo SMS software.

2.3.7.4.6 Redistribute Vertex

Summary

This tool is intended to redistribute the vertex distance of polyline or polygon layers that could be used to create the triangular element mesh of the model. The procedure extracts the vertex of the polyline|polygon features- and execute a routine that uses both sine and cosine theorems, as well as the equation of the line and slope to redistribute the vertex. The routine maintains the connection between different feature parts, for example with a polyline layer the polylines maintain connected between them. Finally, a new polygon or polyline layer is generated.

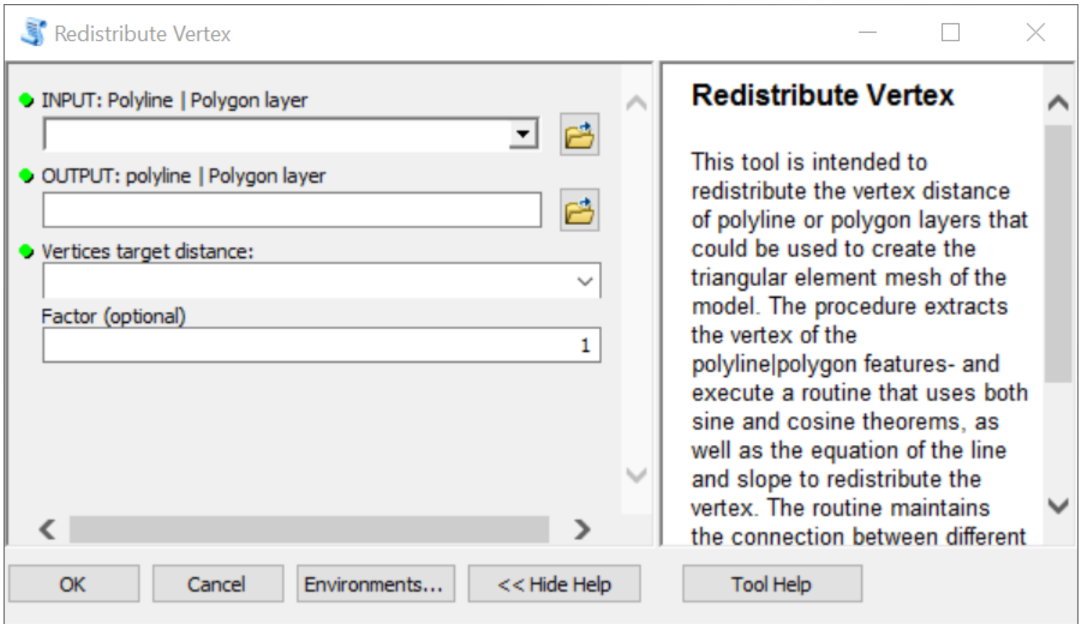


Figure 65: Tool to redistribute the distance between vertex of polyline or polygon layer.

Usage of the tool (Figure 65)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Select the polyline or polygon layer with the target distance to redistribute in the attribute table of the layer.
- Define the output path and namefile of the polyline|polygon layer.
- Select the field where the target distance of vertex is defined.
- Define if the target distance needs a factor to increase or decrease their value.

2.3.7.4.7 Topographic Depressions Evaluation

Summary

Use this tool to fill the DEM and by difference between the filled DEM and the original DEM obtain the depression polygons. Use the depression polygons to define which one should be filled and which ones not with the field IsSink.

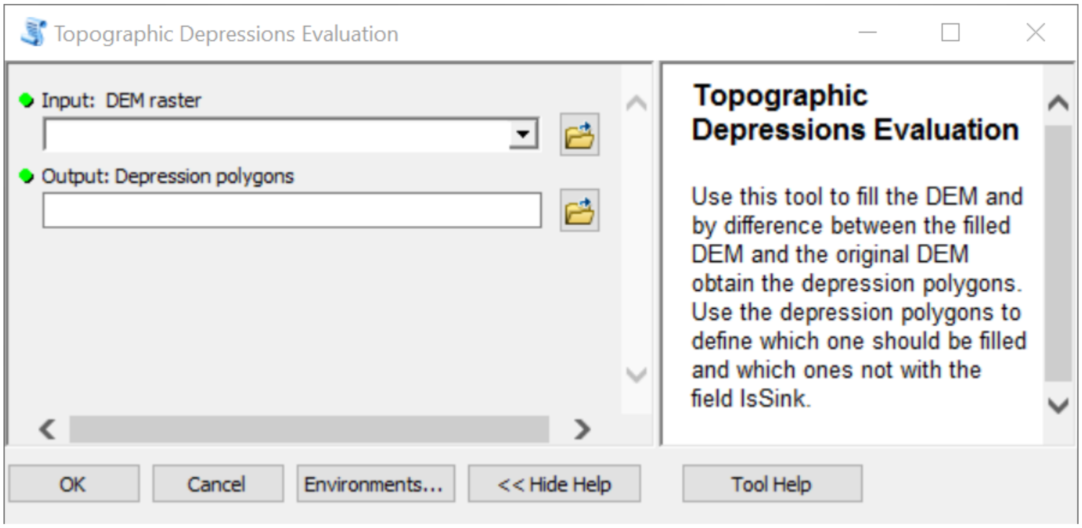


Figure 66: Tool to generate a polygon layer with all the depressions evaluated by the tool.

Usage of the tool (Figure 66)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Run this tool to generate the depression polygon.
- Define which polygons are sinks that should be filled and which ones not. To do so, fill the field IsSink with the value 1 to define the polygons that are sinks and should not be filled, and the value 0 to define artificial sinks that should be filled.

2.3.7.5 Subfunctions

2.3.7.5.1 Delete Extra Fields

Summary

This tool removes the columns that are not in the source layers 1 and 2 from the target layer. These extra columns are generated automatically by some geoprocessing tools used in intermediate processes. This tool is intended to be used as a function inside other scripts.

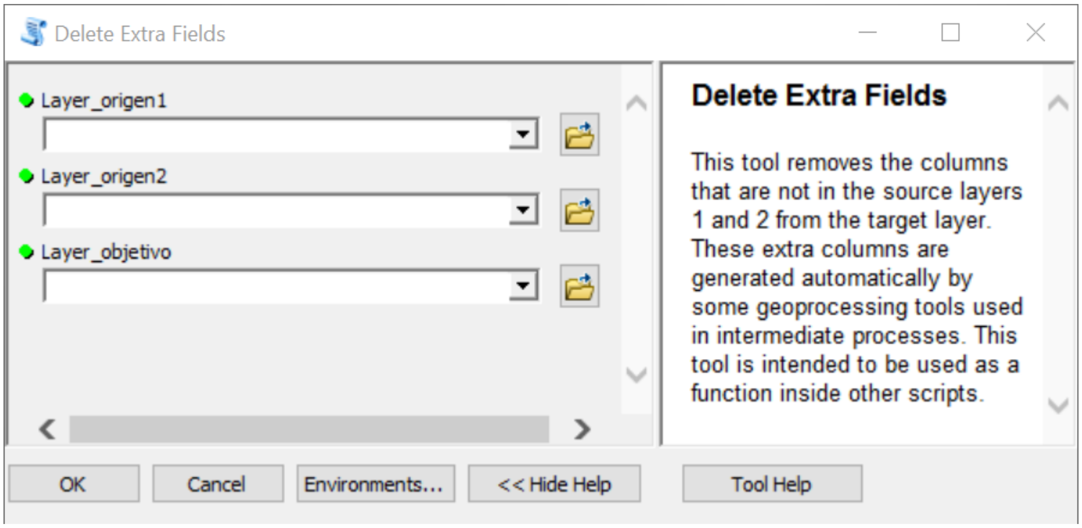


Figure 67: Tool to delete the Extra Columns generated by some geoprocessing tools in the output layer that are not in the source layers.

Usage of the tool (Figure 67)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Identify the source layers 1 and 2 used to create the target layer and select them.
- Select the target layer to delete extra columns that are not in the attribute table of source layers 1 and 2.
- Run the tool and check the attribute table of the target layer. NOTE: this sub-function is intended to be used in intermediate processes of scripts codes and automatically delete extra columns.

2.3.7.5.2 Update Field by Raster

Summary

Use this tool to update a point layer attribute table field with information of a raster layer.

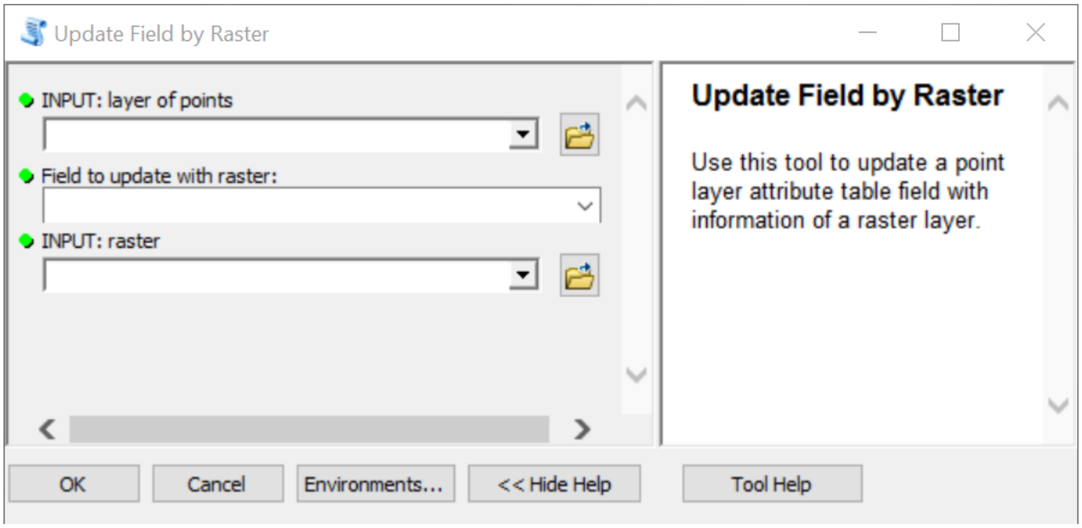


Figure 68: Tool that update a field of a point layer with the information from a raster layer.

Usage of the tool (Figure 68)

FOLLOW THESE STEPS TO USE THIS TOOL:

- Select a layer of points to update a field of the attribute table by raster.
- Select the field to update, consider a field numeric type.
- Select the raster to update the selected field.

MELEF FSW HELP MANUAL

Numerical Hydrological Model for
the integrated simulation of regional
groundwater and surfacewate flows
in continental and coastal
watersheds.

Part III

3 TECHNICAL MANUAL

3.1 MELEF FSW

3.1.1 Input simulation files

In this section you will find a description of the different input simulations files that MELEF FSW can use to approximate the continental and coastal hydrology in a catchment level.

The input simulation files described in the following sections are: COR; CND; ELE; INI; INP; PRN; SEC; SOI; SLC; SLR.

3.1.1.1 Input COR file

This file stores the coordinates X and Y of the mesh nodes:

The following is an example of the first lines of a COR file:

```
43278        2
1   567130.8820  2674098.4500
2   567215.5240  2674152.3700
3   567297.6370  2674210.0700
4   567379.4070  2674268.2500
5   567461.1760  2674326.4400
6   567542.9450  2674384.6300
7   567624.7140  2674442.8200
8   567706.4830  2674501.0100
```

The printing format used for the COR files is the following:
Header: `'%10g%10g\n' [TotalNudos; ColumnasInfo]`
Information: `'%6g %13.4f %13.4f\n' [NNUDO; CoorX; CoorY]`

3.1.1.2 Input CDN file

This file stores the boundary conditions on the skin of the mesh to impose a groundwater level value or a gradient value. These values can be constant or transient to adjust a temporal evolution.

The following is an example of a CND file with a transient evolution of water level imposed that start in 1955 meters and finish in 1952 meters for a starting time 0 and end time 172800 seconds (2 days).

```
           0
1000000000            1.9550E+03
   348   349   350   351   352   353   354   355   356   357     0     0     0
0


     86400
```

```
1000000000            1.9530E+03
   348   349   350   351   352   353   354   355   356   357     0     0     0
0


    172800
1000000000            1.9520E+03
   348   349   350   351   352   353   354   355   356   357     0     0     0
0
```

The printing format used for the CND files is the following:
Time: `'%10f\n'` `[TimeSeconds]`
Code of use and imposed value: `%10d%22.4E\n'` `[COD_USE; VALUE]`
Time: `'%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g% 6g\n'` `[NNUDO(1 X 13)]`

NOTE: The file first prints the time in seconds, followed by the code of use that defines if the boundary condition is for continental or coastal hydrology and the imposed value, the next is a list of all the nodes with this boundary condition, and finally a zero value to finish this time and start a new one if required.

3.1.1.3 Input ELE file

This file stores the node numbers that form each triangular element of the mesh. It is mandatory that the elements are numbered counterclockwise in order to run the MELEF FSW finite element model with no problems.

The following is an example of the header and the first seven elements of an ELE file:

```
     86206         3
       348       349         1
       350         1       349
       350         2         1
         2       350       351
       351         3         2
       352         3       351
       353         3       352
```

The printing format used for the ELE files is the following:
Header: `'%10g%10g\n'` `[TotalElements; NodesbyElement]`
Information: `'%10g %10g %10g\n'` `[NodeVertex1; NodeVertex2; NodeVertex3]`

3.1.1.4 Input INI file

This file stores the initial solution for each node of the mesh. The printing order of the nodes is ascending, so the row position minus the header implies mesh node number.

The following is an example of the initial solution used in a continental fresh water simulation, the initial solution file has a header followed by the fresh water level of all the nodes of the mesh. The fresh water level is referred to the mean sea water level and if this value is over the topography then there is surface water or by contrary only groundwater.

```
48274         1    76204800.000000
   1955.4200000000001
   1955.5300000000000
   1955.6600000000001
   1955.7900000000000
   1956.4300000000001
   1958.4200000000001
   1961.9000000000001
```

The printing format of the INI files for continental simulation is the following:
Header: `'%10g%10g%19f\n' [TotalNodes; FreedomDegrees; TimeSeconds]`
Information: `'%19.13f\n' [FreshWaterLevel]`

When the degrees of freedom is one, then the solution belongs only to a continental flow simulation, but when it is two, then the solution belongs to a simulation with continental and coastal interaction: there in the first column is printed the position of the water table (fresh) and in the second column the position of the immiscible interface of the saltwater (saline wedge) as is showed in the following example.

```
42342          2    157766400.000000
   0.1757019951337        2.5524728690084
   1.0063108178119       -1.9746104201661
   2.6399649363506      -30.6493182246236
   4.2173824645556      -73.4324037656311
   5.6311015351218     -135.4947109446500
```

The printing format of the INI files for continental and coastal simulation is the following:
Header: `'%10g%10g%19f\n' [TotalNodes; FreedomDegrees; TimeSeconds]`
Information: `'%19.13%19.13f\n' [FreshWaterLevel; SaltWaterLevel]`

3.1.1.5 Input INP file

File that stores all the configuration of the simulation execution, as well as the configuration of the global parameters.

This is a sample of the file contents:

```
#
#----------------------------------------------------------------------
# MALLAJE DE TRIANGULOS LINEALES TRIANG-1 A 2 DIMENSIONES:  (T3)
```

```
#                    CUENCA DEL RIO BARCES
#            Elemento con solucion para H (ACUIL-2H)
#               Fichero del elemento:
#       Tipo de elemento: T3 lineal … 2 dimensiones
#
#                    PROBLEMA TRANSITORIO/PERMANENTE
#
#         Mallaje (N elementos: ; N nudos: )
#Condiciones: IMPUESTA(descarga), Interna(P-E-Bombeos-Derivaciones)
#---------------------------------------------------------------------
#
!---NAME OF INPUT/OUTPUT FILES
_MFIL = 'MELW9'


!---OUTPUT FILES TO PRINT: ACTIVATE/INACTIVE (!)
!_MFIN = '.fin'
_MPST = '.vno'
_MSEC = '.sec'
_MDEB = '.deb'
_MVEL = '.vel'
_MSOL = '.sol'


!---TYPE OF ELEMENT
_ELTYP = 'AL2H'


!---TOTAL EXECUTION WORK
_MEXE = '.eta'
_NEXE = 8748000


!---TRANSIENT RESOLUTION.
_DPAS = 7200
_NPAS = 8748
_ALFA = 0.5


!---PRINTING REUSLTS
_IFPST = 12
_IFSEC = 2
!_IFFIN = 12
_IFVEL = 36
_IFSOL = 12


!---READING LEAP: DEFINE TIME (SECONDS)
!---TINI GLOBALLY CONTROLS (EXCEPT INI) THE READING LEAP:
_TINI = 63158400
!_TASCND = 63158400
!_TASPRN = 63158400
!_TASSOI = 63158400
!_TASSLR = 63158400
!_TASSLC = 63158400


!---READING GEOMETRY
_MCOR = '.cor'
COOR
_MELE = '.ele'
ELEM
PEAU


!---GLOBAL PROPERTIES PRGL
!-(F-Ghy,Pr,FE-alv,Coef-X-Superf,Esp-Min,Esp-Cap,ETP,EP,%RAU|CCAMPO_PM,K-Min-
Inter,F_Galeria,QDet,VDetMax,EspRaiz,+Qdesagüe,+Hdesagüe,+hcompuerta)
PRGL(0,0,-100,-0.936,3,.5,3.1e-11,0,.1,2e-7,1,0,1.0e+6,5,0,0,0)


!---INPUT FILES READING
_MCND = '.cnd'
```

```
COND
_TASINI = 76204800
_MINI = '.ini'
INIT()
_MPRN = '.prn'
PRNO
_MSOI = '.soi'
SOIL
_MSLR = '.slr'
SOLR
!_MSLC = '.slc'
!SOLC

!--- SYSTEM RESOLUTION
_EPSDL = 1.0E-15
_NPREC = 2
_NRDEM = 20
```

3.1.1.6 Input PRN file

This file stores properties of geological materials and topography elevation.

This is an example of the header and properties of the first five nodes of the mesh:

```
    48274         8         0
 3.0000E-03  1.98598E+03  1.92336E+03  8.101852E-07  8.101852E-07  0.00E+00  8.101852E-07  1.98598E+03
 3.0000E-03  1.98640E+03  1.92556E+03  8.101852E-07  8.101852E-07  0.00E+00  8.101852E-07  1.98640E+03
 3.0000E-03  1.98817E+03  1.92783E+03  8.101852E-07  8.101852E-07  0.00E+00  8.101852E-07  1.98817E+03
 3.0000E-03  1.98833E+03  1.92988E+03  8.101852E-07  8.101852E-07  0.00E+00  8.101852E-07  1.98833E+03
 3.0000E-03  2.00089E+03  1.93159E+03  8.101852E-07  8.101852E-07  0.00E+00  8.101852E-07  2.00089E+03
```

The printing format used for the PRN files is the following:
Header: `'%10d%10g%10.0f\r\n' [TotalNudos; ColumnasParam; TiempoSegundos]`
Information: `'%11.4E%13.5E%13.5E%14.6E%14.6E%10.2E%14.6E%13.5E\r\n' [N; COTA2; SUSTRATO; KX; KY; ALFA; KZ; COTA1]`

Where the 8 PRN properties are:
N: porosity of materials per unit.
COTA2: secondary elevation of nodes to define the presence of Tunnels that is activated when COTA2 is lower than COTA1.
SUSTRATO: position of the impervious substratum that defines the aquifer base.
KX: hydraulic conductivity of the materials on the x axis in units m/sec.
KY: hydraulic conductivity of the materials on the y axis in units m/sec.
ALFA: Anisotropy angle, in units of degrees, of the hydraulic conductivity, for the X and Y axes, measured from north and counterclockwise.
KZ: Hydraulic conductivity of materials on the z axis, it is interpreted as the infiltration capacity, in units of m/sec.
COTA1: position of primary topography in meters.

3.1.1.7 Input SEC file

This file stores the nodes that conform a gauging sections drawn by the user in a GIS software.

This is an example of the header and the first four sections of 12 sections in a SEC file:

```
    12
     1           0
 25714 25715 24774 24775 23803 22777 21724 21725 20663 20664 20665     0     0
     2           0
 20665 20666 19560 18405 17252 17253 16079 14914 13740 12575 11413     0     0
     3           0
 11413 11414 10258  9047  7823  9050  9051  9052  9053  9054 10271 10272 11438
 12612 13792     0     0     0     0     0     0     0     0     0     0     0
     4           0
 13792 14986 16166 15036 13842 12666 11499 10343  9141  7927     0     0     0
```

The printing format used for the SEC files is the following:
Header: `'%16d\n' [TotalSections]`
Section ID and value: `'%6d%12d\n' [NúmeroSección; Valor]`
Nodes in each section: `'%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g\n' [NNUDO(1 x 13)]`

3.1.1.8 Input SOI file

This file stores the configuration of soil properties, the transient variables of potential evaporation and transpiration, as well as of the hydraulic conductivity of the surface environment.

The SOI file is divided in two sections, the firs section has all nodes and eight columns of information as is showed in the following example with the first lines of the file:

```
     43278         8         1
 11    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 11    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 11    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 11    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 11    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 12    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 12    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
 12    5.000E-01  0.000000E+00  0.000000E+00  1.000E+00  1.000E-04  0.000E+00  3.500E-01
```

The printing format of the section 1 of the SOI file is the following:
Header: `'%10d%10g%10.0f\n' [TotalNodes; Columns; TimeSeconds]`
Information: `'% -5d % -1.4E % -1.6E % -1.6E % -1.4E % -1.4E % -1.2E % -1.4E\r\n' [ZONA; EC; ETP; EP; ES; KSUPERF; SLOPE; CCAMPO_PM]`

This first section has eight columns `[ZONE; EC; ETP; EP; ES; KSUPERF; SLOPE; CCAMPO_PM]` whit values obtained from the SOIMELEF layer generated with the GIS Toolbox, the first column is the zone or identification number that is important to connect with the second section of the file that has the transient behavior of the defined columns.

Where the 7 soil properties are:
EC: thickness of the capillary fringe of the soil in meters.
ETP: potential evapotranspiration rate in m/sec.
EP: potential evaporation rat in m/sec.
ES: soil thickness, as regards the maximum plants roots length.
KSURF: hydraulic conductivity of the superficial medium in m/sec.
SLOPE: ground slope in degrees.
CCAMPO_PM: Soil field capacity minus the wilting point in %.

Section 2 of the simulation file SOI:

```
  371        25        12      0110000
     0  4.398148E-08 4.178241E-08 3.935185E-08 3.738426E-08 3.935185E-08 3.738426E-08 ...
 86400  4.398148E-08 4.178241E-08 3.935185E-08 3.738426E-08 3.935185E-08 3.738426E-08 ...
172800  4.513889E-08 4.288194E-08 3.819444E-08 3.628472E-08 4.050926E-08 3.848380E-08 ...
259200  4.745370E-08 4.508102E-08 3.703704E-08 3.518519E-08 4.282407E-08 4.068287E-08 ...
345600  4.861111E-08 4.618056E-08 3.472222E-08 3.298611E-08 4.513889E-08 4.288194E-08 ...
432000  4.745370E-08 4.508102E-08 5.324074E-08 5.057870E-08 4.513889E-08 4.288194E-08 ...
518400  4.745370E-08 4.508102E-08 4.513889E-08 4.288194E-08 4.282407E-08 4.068287E-08 ...
```

The second section has a second header with four columns:
Column1: number of lines with the time of transient behavior `[371]`
Column2: total columns with information `[25]` where the first column is time.
Column3: number of zones that has the SOIMELEF file `[12]`
Column4: 7 digit code that represent the 7 properties of the SOI file. For example, the code `[0110000]` defines that only the properties [2, 3] have a transient behavior and and the properties [1, 4, 5, 6, 7] are constant and their value is defined in the first section of this file.

3.1.1.9 Input SLC file

This file stores the Stream Lateral boundary Condition (SLC) that is used to define stream inputs by the mesh skin.

This is a sample of the file printing, where after the time are printed the MELEF usage code and the value of the simulation condition with exponential notation, and later the nodes where the condition is applied:

```
         0
1000000000           -1.32000E-3
   392   393   394   395   396   397   398   399   400   401   402   403   404
   405   406   407     0     0     0     0     0     0     0     0     0     0
0
     86400
1000000000           -1.3300E-3
   392   393   394   395   396   397   398   399   400   401   402   403   404
   405   406   407     0     0     0     0     0     0     0     0     0     0
0
```

The printing format used for the SLC files is the following:
Time: `'%10f\n' [TimeSeconds]`
Code of use and flow value: `'%10d%22.4E\n' [COD_USE; VALUE]`
Nodes with a SLC condition: `'%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g% 6g\n' [NNUDO(1 X 13)]`

The described structure is printed for each new time.

3.1.1.10 Input SLR file

This file stores the distributed water uses (pumping, injection, diversions, others), as well as the diffuse surface water inputs (precipitation-irrigation).

The SLR file is printed in two sections when it is a transient file and in one section when it is a permanent file.

The first section is a common section for transient and permanent SLR file, in this section is printed the water use MELEF codes, the permanent water use value and the node numbers that define the zones.

The following is an example of a permanent SLR file, where:
- In the first line: The time is printed (equal to zero) and a code to identify that the second section was not printed (equal to zero).
- In the second line: The MELEF code of use is printed followed by its value in scientific notation in basic units (see basic units).
- In the third line: The number of nodes that are included in the zone are printed.

The second and third lines are printed repeatedly for each zone contained in the SLRMELEF.shp file prepared with the GIS Toolbox.

```
          0              0
```

```
1000000000           3.1688E-04
 17113     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           6.3376E-04
 18112     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           6.3376E-04
 20140     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           6.3376E-04
 15107     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           9.5064E-04
-19145-20151-21133-21134-22108-22109-22110-23088-23089-23090-23091-24020-24021
-24022-24023-24024-24901 18122     0     0     0     0     0     0     0     0
1000000000           1.9013E-03
-17121-18128-18130-18131-19146-19147-19148-19149-20152-20153-20154-20155-21136
-21137-21138-22111-22112-22113-22114-23093-23094-23095-23096-24025-24026-24027
-24028-24029-24909-24910-25755-26593-26594-26595-26596-27413 18124     0     0
.
.
.
2000000000           0.0000E+00
 26826 26827 26828 26829 26830 26831 27636 27637 27638 27639 27641 27642 27643
 27644 28423 28425 28426 28427 28428 28429 28430 28431 28432 29194 29195 29197
 29198 29199 29200 29201 29202 29203 29204 29949 29950 29951 29954 29955 29956
 29957 29958 29959 29960 30704 30705 30706 30711 31435 31436 31437 31438 31439
 31440 31441 32160 32161 32163 32164 32165 32166 32167 32881     0     0     0
2000000000           0.0000E+00
 33547 33548 33549 33550 33551 34195 34196 34197 34200 34201 34203 34205     0
0
```

The following is an example of a transient SLR file, where:
- First line: The time is printed (equal to zero) and a code to identify that the second section with transient values is printed (equal to one).
- Second and third lines: See previous description for the example of a permanent SLR file.

The second and third lines are printed repeatedly for each zone contained in the SLRMELEF.shp file prepared with the GIS Toolbox.

```
         0         1
1000000000           3.1688E-04
 17113     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           6.3376E-04
 18112     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           6.3376E-04
 20140     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           6.3376E-04
 15107     0     0     0     0     0     0     0     0     0     0     0     0
1000000000           9.5064E-04
-19145-20151-21133-21134-22108-22109-22110-23088-23089-23090-23091-24020-24021
-24022-24023-24024-24901 18122     0     0     0     0     0     0     0     0
1000000000           1.9013E-03
```

```
-17121-18128-18130-18131-19146-19147-19148-19149-20152-20153-20154-20155-21136
-21137-21138-22111-22112-22113-22114-23093-23094-23095-23096-24025-24026-24027
-24028-24029-24909-24910-25755-26593-26594-26595-26596-27413 18124     0     0
.
.
.
2000000000            0.0000E+00
 26826 26827 26828 26829 26830 26831 27636 27637 27638 27639 27641 27642 27643
 27644 28423 28425 28426 28427 28428 28429 28430 28431 28432 29194 29195 29197
 29198 29199 29200 29201 29202 29203 29204 29949 29950 29951 29954 29955 29956
 29957 29958 29959 29960 30704 30705 30706 30711 31435 31436 31437 31438 31439
 31440 31441 32160 32161 32163 32164 32165 32166 32167 32881     0     0     0
2000000000            0.0000E+00
 33547 33548 33549 33550 33551 34195 34196 34197 34200 34201 34203 34205     0
0
```

The following is an example of a second section in a transient SLR file, where:
- In the first line: The number of rows with temporal information is printed (8401), followed by the number of columns (number of zones plus one column with time information).
- In the second line: The time in seconds is printed in the first column, followed by n columns that has the water use value of each zone (the order of zones printed in the first section is the same order as the columns printed in this section).

```
     8401       339

        0  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
    86400  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
   172800  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
   259200  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
   345600  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
   432000  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
   518400  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
.
.
.
725500800  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
725587200  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
3.1710E-04  2.5368E-03 ...
```

```
725673600  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
 3.1710E-04  2.5368E-03 ...
725760000  3.1710E-04  3.1710E-04  3.1710E-04  9.5129E-05
 3.1710E-04  2.5368E-03 ...
```

The printing format used for the first section of the SLR file is the following:

line 1: `'%10.0f%10d\r\n' [Time in seconds, 0 or 1]`

line 2: `'%10d%22.4E\r\n' [Cod of use; Water use value]`

line 3: `'%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g%6g\r\n' [NNUDO(1 X 13)]`

The printing format used for the second section is the following:

line 1: `'%10d%10d\r\n' [Rows printed, Columns printed]`

line 1: `'%10d % -1.4E %-1.4E ... % -1.4E\r\n' [Time in seconds, Value Zone 1, Value Zone 2, ..., Value Zone n]`

3.1.2 Result simulation files

This section describes the result simulation files of the MELEF FSW model.

The output simulation files described in the following sections are: DEB; ETA; FIN; SOL; VEL; VNO.

3.1.2.1 Output DEB file

This file stores the water flow that pass through the gauging sections defined in the simulation SEC file. The printed columns have information about the water flow, water velocity, and variations of flow (uncertainty).

This is a sample of the information contained in the DEB file for 13 gauging sections with 11 parameters for the continental freshwater model:

```
 13   11   63172800.
  1 6.23704E-05 2.31559E-04 0.00000E+00 1.04898E-06 4.48761E-06 0.00000E+00 2.88610E-05 7.97545E-05 0.00000E+00 1.73101E-05 1.73101E-03
  2 1.89885E-03 6.08993E-03 0.00000E+00 3.53185E-05 1.39823E-04 0.00000E+00-7.51359E-04-3.23936E-03 0.00000E+00 5.03351E-04 5.03351E-02
  3 8.64595E-04 1.20937E-03 0.00000E+00 1.27435E-05 2.19480E-05 0.00000E+00-9.43976E-05-2.13314E-04 0.00000E+00 3.29004E-04 3.29004E-02
  4-8.81274E-05 4.97268E-06 0.00000E+00-4.13617E-07 2.79314E-08 0.00000E+00 1.59383E-04 2.54077E-05 0.00000E+00-6.29549E-05-3.56659E-03
  5 2.11137E-05 2.03674E-04 6.77726E-04 3.63628E-07 4.25524E-06 5.59706E-05 2.97890E-06 3.20291E-04 1.77263E-04 7.87039E-06 7.87039E-04
  6 9.27071E-04 2.38062E-04 0.00000E+00 3.20980E-06 1.13679E-06 0.00000E+00 4.12180E-04 1.14530E-04 0.00000E+00 3.17342E-04 3.17342E-02
  7 4.31259E-05 3.45283E-03 2.80028E-02 9.76531E-07 9.46705E-05 8.28852E-04 1.28777E-05 1.96796E-03-2.55466E-03 2.38081E-05 2.38081E-03
  8 1.04095E-04 3.01647E-05 0.00000E+00 1.58260E-06 5.55511E-07 0.00000E+00 2.53388E-05-7.79487E-06 0.00000E+00 4.91695E-05 4.91695E-03
  9 5.24703E-06 9.56670E-05 9.82565E-04 1.37587E-07 2.49673E-06 8.78090E-05 1.06664E-06 7.01221E-05 1.30424E-03 2.40613E-06 2.40613E-04
 10-2.03631E-04 6.84597E-05 0.00000E+00-9.60610E-07 3.69122E-07 0.00000E+00 7.24924E-05 1.70116E-04 0.00000E+00-2.02094E-04-1.37501E-02
 11-9.53300E-04-1.16830E-02 0.00000E+00-1.29643E-05-1.90465E-04 0.00000E+00-9.97126E-04-1.93868E-02 0.00000E+00-1.19452E-03-1.19452E-01
 12 5.99973E-03 2.91941E-02 4.40039E-02 2.28355E-04 1.02578E-03 4.40039E+00-3.36355E-03-2.15294E-02-3.51453E-02 2.71482E-03 2.71482E-01
 13 1.40426E-03 1.22729E-02 8.52120E-02 2.81616E-05 2.88431E-04 5.46693E-02-4.24616E-04-6.01074E-03-4.60596E-02 9.86652E-04 9.86652E-02
```

This is a sample of the information contained in the DEB file for 18 gauging sections with 14 parameters for the coastal and continental freshwater model:

```
18   14      10800.
    1 2.57845E-03 2.74481E-03 0.00000E+00 2.47776E-07 3.32411E-08 0.00000E+00-3.04275E-04-2.97284E-04 0.00000E+00 0.00000E+00 0.00000E+00 3.92823E-03
0.00000E+00 0.00000E+00
    2 5.16467E-03 4.54063E-03 0.00000E+00 8.48035E-07 6.25675E-08 0.00000E+00 2.27262E-04 2.07246E-04 0.00000E+00 0.00000E+00 0.00000E+00 1.29589E-02
0.00000E+00 0.00000E+00
    3 3.37362E-03 2.22798E-03 0.00000E+00 6.48444E-07 1.35673E-08 0.00000E+00 3.98767E-04 2.58772E-04 0.00000E+00 0.00000E+00 0.00000E+00 3.46573E-02
0.00000E+00 0.00000E+00
    4 3.00921E-07 2.59051E-07 0.00000E+00 7.90909E-08 4.54064E-12 0.00000E+00 1.80677E-07 1.55538E-07 0.00000E+00 0.00000E+00 0.00000E+00 6.01842E-06
0.00000E+00 0.00000E+00
    5-1.03987E-03-6.70483E-02 1.17946E-01-3.93951E-07-1.11408E-07 2.25575E-01 2.39865E-02 1.34711E+00 4.61338E+00 0.00000E+00 0.00000E+00-2.04417E-02-
1.07323E+00 2.06144E+00
    6-1.54855E-04-8.17814E-03-2.38994E-02-2.11996E-07-1.39074E-07 0.00000E+00 6.73600E-06 3.41481E-04 9.89727E-04 0.00000E+00 0.00000E+00-2.53951E-03
0.00000E+00 0.00000E+00
    7 3.30091E-03 3.23783E-03 0.00000E+00 3.59786E-07 6.05254E-08 0.00000E+00-5.46522E-03-4.11041E-03 0.00000E+00 0.00000E+00 0.00000E+00 2.40420E-02
0.00000E+00 0.00000E+00
    8 1.16968E-03 9.27329E-04 0.00000E+00 2.38470E-07 4.71346E-08 0.00000E+00-2.74004E-04-1.64696E-04 0.00000E+00 0.00000E+00 0.00000E+00 2.23825E-03
0.00000E+00 0.00000E+00
    9 8.02331E-04 6.56203E-04 0.00000E+00 1.57387E-07 2.01254E-08 0.00000E+00-3.37979E-04-2.06237E-04 0.00000E+00 0.00000E+00 0.00000E+00 1.39234E-03
0.00000E+00 0.00000E+00
   10-8.65727E-03-6.09860E-03 0.00000E+00-5.62287E-07-9.82972E-08 0.00000E+00-2.61089E-03-1.74597E-03 0.00000E+00 0.00000E+00 0.00000E+00-8.97658E-03
0.00000E+00 0.00000E+00
   11-4.96097E-03-4.49575E-03 0.00000E+00-5.33786E-07-6.48944E-08 0.00000E+00-9.49842E-04-8.70521E-04 0.00000E+00 0.00000E+00 0.00000E+00-5.05534E-03
0.00000E+00 0.00000E+00
   12-2.41458E-03-1.58826E-03 0.00000E+00-2.64016E-07-2.67508E-08 0.00000E+00-5.96170E-04-4.30974E-04 0.00000E+00 0.00000E+00 0.00000E+00-3.00737E-03
0.00000E+00 0.00000E+00
   13-1.57038E-03-1.37879E-03 0.00000E+00-8.62101E-08-1.06091E-08 0.00000E+00-8.23242E-04-7.46930E-04 0.00000E+00 0.00000E+00 0.00000E+00-2.49258E-03
0.00000E+00 0.00000E+00
   14-2.28837E-04-7.24531E-04-4.68967E-03-1.95971E-07-1.76819E-07-1.20341E-05-2.28944E-04-1.17187E-03-6.31498E-03 0.00000E+00 0.00000E+00 0.00000E+00
0.00000E+00 0.00000E+00
   15-6.40622E-03-3.34001E-03 0.00000E+00-5.89891E-05-3.08856E-06 0.00000E+00-2.60883E-03-1.65399E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
0.00000E+00 0.00000E+00
   16 2.61330E-05 1.27395E-03 3.33839E-01 6.36887E-08 1.47579E-06 1.75835E-04 5.26874E-06 2.67131E-04 6.31378E-02 0.00000E+00 0.00000E+00 0.00000E+00
0.00000E+00 0.00000E+00
   17-6.82907E-03-3.44403E-03 0.00000E+00-9.35683E-06-6.75440E-07 0.00000E+00-9.03226E-04-5.47787E-04 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
0.00000E+00 0.00000E+00
   18-2.99549E-05-4.95302E-04 0.00000E+00-2.25704E-06-4.11322E-09 0.00000E+00 1.64182E-05 2.69236E-04 0.00000E+00 0.00000E+00 0.00000E+00-5.99099E-04-
3.63447E-03 0.00000E+00
```

Explanation of the header file structure:
Number of sections: number of gauging sections
Number of parameters: Number of parameters printed.
Time in seconds: Printing time of the parameter values.

Explanation of the parameters printed in the file:

1 **Qgroundwater [m3/seg]:** Groundwater flow through the gauging section.
2 **Qsubsoil [m3/seg]:** Subsoil water flow through the gauging section.
3 **Qsurface [m3/seg]:** Surface water flow through the gauging section.
4 **Vel.Groundwater [m/seg]:** Groundwater Darcy velocity.
5 **Vel.Subsoil [m/seg]:** Subsoil water Darcy velocity.
6 **Vel.Surface [m/seg]:** Surface water velocity.
7 **Variation Qgrounwater [m3/seg]:** variation of groundwater flow
8 **Variation Qsubsoil [m3/seg]:** variation of subsoil water flow.
9 **Variation Qsurface [m3/seg]:** variation of surface water flow.
10 **QoverlandFlow [m3/seg]:** Overland water flow through the gauging section.

11 **Vel.OverlandFlow [m/seg]:** Overland water velocity.
12 **Qsalt-groundwater [m3/seg]**: Salt groundwater flow through the gauging section. This result appears when the coastal model is activated.
13 **Qsalt-subsoil [m3/seg]**: Salt subsoil flow through the gauging section.
14 **Qsalt-surface [m3/seg]**: Salt surface water flow through the gauging section.

Consider that parameter values can be positive or negative depending on their direction through the gauging section. This implies that if the section is traced from the right side bank to left (looking to downstream) the positive values are directed downstream and the negative values are directed upstream.

3.1.2.2 Output ETA file

This file stores what happens globally during the simulation, this file registers the reading files, the global parameters, the system convergence and a series of global results in order to keep track of the global evolution of the simulation. ETA file prints 41 variables when it is a continental simulation (AL2H model) or 54 when the simulation considers both continental and coastal models (AL2S model).

The following is only a part of the ETA file related with the parameters printed globally for 41 variables:

```
POST-TRAITEMENT (M= 0)
 ===============
               FICHIER DE POST-TRAITEMENT                = Proye22.vno
               NB DE VALEURS A CALCULER PAR NOEUD (NPOST)=    5


              1  Global Precipitation (-)        (m3/d) =  0.00000E+00
              2  Surface inflow on boundaries(-) (m3/d) =  0.00000E+00
              3  Lateral inflow on boundaries(-) (m3/d) =  0.00000E+00
              4  Water Injections (-)            (m3/d) =  0.00000E+00
              5  Zonal Precipitation (-)         (m3/d) =  0.00000E+00
              6  Zonal Recharge (-)              (m3/d) =  0.00000E+00
              7  TOTAL WATER INFLOW (-)          (m3/d) =  0.00000E+00


              8  Global Evapo(Transpiration) (+) (m3/d) =  3.22425E+03
              9  Zonal Transpiration (+)         (m3/d) =  9.90189E+01
              10 SurfaceWater Diversion (+)      (m3/d) =  1.49909E+04
              11 GroundWater Pumping (+)         (m3/d) =  0.00000E+00
              12 Boundary Out/Inflow (+/-)       (m3/d) =  6.94354E+01
              13 TOTAL WATER OUTFLOW (+)         (m3/d) =  1.83836E+04


              14 Global Precipitation Volume (-)  (hm3) = 0.000000E+00
              15 Surface Inflow Volume (-)        (hm3) = 0.000000E+00
              16 Lateral Inflow Volume (-)        (hm3) = 0.000000E+00
              17 Water Injections Volume (-)      (hm3) = 0.000000E+00
              18 Zonal Precipitation Volume (-)   (hm3) =-1.187817E+00
              19 Zonal Recharge Volume (-)        (hm3) =-2.736303E-01
              20 TOTAL INFLOW VOLUME (-)          (hm3) =-1.187817E+00


              21 GlobalEvapo(Transpiration)Volume + hm3 = 2.917996E-02
              22 Zonal Transpiration Volume (+)   (hm3) = 1.213872E+00
              23 Diverted Water Volume (+)        (hm3) = 1.194346E-01
```

```
24 Pumped Water Volume (+)           (hm3) = 0.000000E+00
25 Boundary Out/Inflow Volume (+/-)  (hm3) = 3.933110E-01
26 TOTAL OUTFLOW VOLUME (+)          (hm3) = 1.755797E+00

27 TOTAL IN/OUTFLOW VOLUME (-/+)     (hm3) = 5.679802E-01
28 Continental Basin Area            (km2) = 6.655715E+01
29 SurfaceWater Area                 (km2) = 7.896938E-01
30 Coastal Basin Area                (km2) = 0.000000E+00
31 OverlandFlow Area                 (km2) = 0.000000E+00

32 OverlandFlow thickness              (m) = 0.000000E+00
33 Global Precipitation thickness (-)  (m) = 0.000000E+00
34 Zonal Precipitation thickness (-)   (m) =-1.784657E-02
35 Zonal Recharge thickness (-)        (m) =-4.111209E-03
36 Evapo(Transpiration) thickness (+)  (m) = 4.384196E-04
37 Zonal Transpiration thickness (+)   (m) = 1.823803E-02
38 TOTAL IN/OUTFLOW THICKNESS (-/+)    (m) = 8.298830E-04

39 Continental SurfaceWater Reserve (hm3) = 2.257099E+00
40 Continental Groundwater Reserve  (hm3) = 0.000000E+00
41 TOTAL CONTINENTAL WATER RESERVE  (hm3) = 2.257099E+00
```

The ETA version with 54 variables is the following:

```
POST-TRAITEMENT (M= 0)
 ===============
               FICHIER DE POST-TRAITEMENT                = TIDES4b.vno
               NB DE VALEURS A CALCULER PAR NOEUD (NPOST)=    5

1  Global Precipitation (-)        (m3/d) =  0.00000E+00
2  Surface inflow on boundaries(-) (m3/d) =  0.00000E+00
3  Lateral inflow on boundaries(-) (m3/d) =  0.00000E+00
4  Water Injections (-)            (m3/d) = -8.40084E+03
5  Zonal Precipitation (-)         (m3/d) = -1.72080E+04
6  Zonal Recharge (-)              (m3/d) = -1.72080E+04
7  TOTAL WATER INFLOW (-)          (m3/d) = -1.72080E+04

8  Global Evapo(Transpiration) (+) (m3/d) =  0.00000E+00
9  Zonal Transpiration (+)         (m3/d) =  0.00000E+00
10 SurfaceWater Diversion (+)      (m3/d) =  0.00000E+00
11 GroundWater Pumping (+)         (m3/d) =  2.88749E+04
12 Boundary Out/Inflow (+/-)       (m3/d) =  0.00000E+00
13 TOTAL WATER OUTFLOW (+)         (m3/d) =  2.88749E+04

14 Global Precipitation Volume (-)   (hm3) = 0.000000E+00
15 Surface Inflow Volume (-)         (hm3) = 0.000000E+00
16 Lateral Inflow Volume (-)         (hm3) = 0.000000E+00
17 Water Injections Volume (-)       (hm3) = 0.000000E+00
18 Zonal Precipitation Volume (-)    (hm3) = 0.000000E+00
19 Zonal Recharge Volume (-)         (hm3) = 0.000000E+00
20 TOTAL INFLOW VOLUME (-)           (hm3) = 0.000000E+00

21 GlobalEvapo(Transpiration)Volume + hm3 = 0.000000E+00
22 Zonal Transpiration Volume (+)    (hm3) = 0.000000E+00
23 Diverted Water Volume (+)         (hm3) = 0.000000E+00
24 Pumped Water Volume (+)           (hm3) = 0.000000E+00
25 Boundary Out/Inflow Volume (+/-)  (hm3) = 0.000000E+00
26 TOTAL OUTFLOW VOLUME (+)          (hm3) = 0.000000E+00

27 TOTAL IN/OUTFLOW VOLUME (-/+)     (hm3) = 0.000000E+00
28 Continental Basin Area            (km2) = 1.460286E+03
29 SurfaceWater Area                 (km2) = 0.000000E+00
```

```
30 Coastal Basin Area              (km2) = 0.000000E+00
31 OverlandFlow Area               (km2) = 0.000000E+00

32 OverlandFlow thickness            (m) = 0.000000E+00
33 Global Precipitation thickness (-)  (m) = 0.000000E+00
34 Zonal Precipitation thickness (-)   (m) = 0.000000E+00
35 Zonal Recharge thickness (-)        (m) = 0.000000E+00
36 Evapo(Transpiration) thickness (+)  (m) = 0.000000E+00
37 Zonal Transpiration thickness (+)   (m) = 0.000000E+00
38 TOTAL IN/OUTFLOW THICKNESS (-/+)    (m) = 0.000000E+00

39 Coastal FreshSurfaceWater Volume (hm3) = 0.000000E+00
40 Coastal SaltSurfaceWater Volume  (hm3) = 0.000000E+00
41 Coastal FreshGroundwater Volume  (hm3) = 0.000000E+00
42 Coastal SaltGroundwater Volume   (hm3) = 0.000000E+00
43 COASTAL FRESH-WATER VOLUME       (hm3) = 0.000000E+00
44 TOTAL COASTAL WATER VOLUMES      (hm3) = 0.000000E+00

45 Continental SurfaceWater Reserve (hm3) = 3.161276E+02
46 Continental Groundwater Reserve  (hm3) = 0.000000E+00
47 TOTAL CONTINENTAL WATER RESERVE  (hm3) = 3.161276E+02

48 FRESH SURFACE-WATER VOLUME       (hm3) = 3.161276E+02
49 FRESH GROUNDWATER VOLUME         (hm3) = 0.000000E+00
50 TOTAL FRESH-WATER VOLUMES        (hm3) = 3.161276E+02
51 TOTAL SALT-WATER VOLUMES         (hm3) = 0.000000E+00
52 SURFACE-WATER RESERVE            (hm3) = 3.161276E+02
53 GROUNDWATER RESERVE              (hm3) = 0.000000E+00
54 TOTAL WATER RESERVE              (hm3) = 3.161276E+02
```

The global results of both versions of the ETA file and their meaning are described below:

AL2H	AL2S	Description
1	1	**Global Precipitation (-) (m3/day):** Rainfall intensity imposed globally in the INP file (User Interface).
2	2	**Surface inflow on boundaries (-) (m3/day):** Lateral contribution of rivers or surface water transfer imposed in the CND file (GIS Toolbox).
3	3	**Lateral inflow on boundaries (-) (m3/day):** Lateral diffusive contribution of groundwater imposed in the CND file (GIS Toolbox).
4	4	**Water Injections (-) (m3/day):** Total water flows injected (water spills) imposed in the SLR file (GIS Toolbox).
5	5	**Zonal Precipitation (-) (m3/day):** Precipitation intensity imposed zonally in the SLR file (GIS Toolbox).
6	6	**Zonal Recharge (-) (m3/day):** Vertical water recharge to the aquifer (internally evaluated).
7	7	**TOTAL WATER INFLOW (-) (m3/day):** Total summary of contributions (items 1 to 5) from rivers, lateral diffusive, zonal surface water and rainfall (internally evaluated).
8	8	**Global Evapo(Transpiration) (+) (m3/day):** Global evaporation intensity (+transpiration in transition zones of surface/ground water)

		(internally evaluated). Potential value imposed globally in the INP file (User Interface).
9	9	**Zonal Transpiration (+) (m3/day):** Zonally evaluated transpiration intensity (internally evaluated). Potential value imposed zonally in the SOI file (GIS Toolbox).
10	10	**SurfaceWater Diversion (+) (m3/day):** Evaluated surface water diversion flow rate (global and/or zonal) (internally evaluated). Potential value imposed globally in the INP file and zonally in the SLR file (User Interface and GIS Toolbox).
11	11	**GroundWater Pumping (+) (m3/day):** Groundwater withdrawal flows imposed zonally in the SLR file (GIS Toolbox).
12	12	**Boundary Out/Inflow +/- (m3/day):** Summary of exits/entries at open boundaries (internally evaluated).
13	13	**TOTAL WATER OUTFLOW (+) (m3/day):** Summary of total exits of items 8 to 12 (internally evaluated).
14	14	**Global Precipitation Volume (-) (hm3):** Global Volume of rainfall (cumulative value).
15	15	**Surface Inflow Volume (-) (hm3):** Volume of lateral contribution to rivers or surface water (cumulative value).
16	16	**Lateral Inflow Volume (-) (hm3):** Volume of lateral diffusive contribution to groundwater (cumulative value).
17	17	**Water Injections Volume (-) (hm3):** Volume of water injections to the aquifer (cumulative value).
18	18	**Zonal Precipitation Volume (-) (hm3):** Volume of surface zonal precipitation (cumulative value).
19	19	**Zonal Recharge Volume (-) (hm3):** Volume of vertical recharge to the aquifer (cumulative value).
20	20	**TOTAL INFLOW VOLUME (-) (hm3):** Total volume of entries/inflows (cumulative value).
21	21	**Global Evapo(Transpiration) Volume (+) (hm3):** Volume of global evaporation (+transpiration in transition zones of surface/ground water) evaluated from surface water bodies (accumulated value).
22	22	**Zonal Transpiration Volume (+) (hm3):** Zonally evaluated vegetation transpiration volume (cumulative value).
23	23	**Diverted Water Volume (+) (hm3):** Volume of surface water diverted (cumulative value).
24	24	**Pumped Water Volume (+) (hm3):** Volume of groundwater pumping extraction (cumulative value).

25	25	**Boundary In/Outflow Volume +/- (hm3):** Volume of surface/ground water exits/entries at open boundaries (cumulative value).
26	26	**TOTAL OUTFLOW VOLUME (+/-) (hm3):** Balance of all the exits/outflows volumes (cumulative value).
27	27	**TOTAL IN/OUTFLOW VOLUME (-/+) (hm3):** Balance of all entries/exits volume (item 20+ item 26) of water.
28	28	**Continental Basin Area (km2):** Continental surface of the discrete model.
29	29	**SurfaceWater Area (km2):** Evaluated area of surface water bodies.
30	30	**Coastal Basin Area (km2):** Evaluated area of coastal seawater.
31	31	**OverlandFlow Area (km2):** Evaluated area of existing overland flow.
32	32	**OverlandFlow thickness (m):** Thickness of overland water zonally evaluated in m^3/m^2 of soil surface.
33	33	**Global Precipitation thickness (-) (m):** Precipitation thickness imposed globally in the INP file.
34	34	**Zonal Precipitation thickness (-) (m):** Precipitation thickness imposed zonally in the SLR file.
35	35	**Zonal Recharge thickness (-) (m):** Zonally evaluated vertical recharge thickness to the aquifer.
36	36	**Evapo(Transpiration) thickness (+) (m):** Evaporation thickness evaluated zonally from surface water bodies (+transpiration in transition zones of surface/ground water)
37	37	**Zonal Transpiration thickness (+) (m):** Zonally evaluated vegetation transpiration thickness.
38	38	**TOTAL IN/OUTFLOW THICKNESS (-/+) (m):** Balance of all input/output of water thickness (sum of items 33, 34, 36, 37).
	39	**Coastal FreshSurfaceWater Volume (hm3):** Volume of fresh surface water evaluated in the coastal area.
	40	**Coastal SaltSurfaceWater Volume (hm3):** Volume of salt surface water evaluated in the coastal zone.
	41	**Coastal FreshGroundwater Volume (hm3):** Volume of fresh groundwater evaluated in the coastal zone.
	42	**Coastal SaltGroundwater Volume (hm3):** Volume of salt groundwater evaluated in the coastal zone.
	43	**COASTAL FRESH-WATER VOLUME (hm3):** This variable sums the fresh surface-water and fresh groundwater volumes evaluated in the

coastal zone.
TOTAL COASTAL WATER VOLUMES (hm3): This variable sums the
44 volume of freshwater and saltwater, both surface-water and
groundwater, evaluated in the coastal zone.

39 45 **Continental SurfaceWater Reserve (hm3):** Volume of fresh surface-water evaluated in the continental zone.

40 46 **Continental Groundwater Reserve (hm3):** Volume of fresh groundwater evaluated in the continental zone.

TOTAL CONTINENTAL WATER RESERVE (hm3): This variable sums
41 47 the volume of freshwater, both surface-water and groundwater,
evaluated in the continental zone.

48 **FRESH SURFACE-WATER VOLUME (hm3):** This variable sums the volume of freshwater in the continental and coastal zones.

49 **FRESH GROUNDWATER VOLUME (hm3):** This variable sums the volume of groundwater in the continental and coastal zones.

TOTAL FRESH-WATER VOLUMES (hm3): This variable sums the
50 volume of freshwater, both surface-water and groundwater, evaluated
in the continental and coastal zones.

TOTAL SALT-WATER VOLUMES (hm3): This variable sums the
51 volume of saltwater, both surface-water and groundwater, evaluated
in the coastal zone.

SURFACE-WATER RESERVE (hm3): This variable sums the volume of
52 surface-water, both freshwater and saltwater, evaluated in the
continental and coastal zones.

GROUNDWATER RESERVE (hm3): This variable sums the volume of
53 groundwater, both freshwater and saltwater, evaluated in the
continental and coastal zones.

TOTAL WATER RESERVE (hm3): This variable sums the volume of
54 surface-water and groundwater, both freshwater and saltwater,
evaluated in the continental and coastal zones.

3.1.2.3 Output FIN file

File that stores the solution of the phreatic levels for each node of the mesh (first column), as well as the position of the immiscible interface with seawater whether it is solving coastal flow (second column). The purpose of this file is to serve as an initial solution for the MELEF model by changing the extension from .FIN to .INI.

This is a sample of the information contained in a FIN file with only a degree of freedom (continental freshwater flow). It can have two degrees of freedom, thence a second column with the location of the saltwater wedge or fresh/salt immiscible interface.

```
22014        1    55209600.000000
 35.5858000000000
 39.1953000000000
 41.3374000000000
 42.4534000000000
 47.4867000000000
```

Were:
Header: number of nodes; degree of freedom (1) if continental flow and (2) if both continental (freshwater) and coastal (seawater) flows; and time (sec).
Followed by the columns:

1. **Phreatic levels in nodes:** location of phreatic levels in every node of the numerical mesh (meters).
2. **Immiscible interface:** location of the immiscible fresh/saltwater interface in every node of the numerical mesh (whether considered/solved in coastal problems).

3.1.2.4 Output SOL file

This file stores nodal water variables and transient results of the model.

This is a sample of the information that is printed in the SOL file:

```
48274    8   63244800.
    1 0.00000E+00-3.54666E-06 0.00000E+00 5.07112E-06 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
    2 0.00000E+00-6.24748E-06 0.00000E+00 8.93299E-06 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
    3 0.00000E+00-6.20798E-06 0.00000E+00 8.87704E-06 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
    4 0.00000E+00-5.83970E-06 0.00000E+00 8.35190E-06 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
    5 0.00000E+00-5.14504E-06 0.00000E+00 7.36604E-06 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
    6 0.00000E+00-8.29545E-06 0.00000E+00 1.18796E-05 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
    7 0.00000E+00-1.29269E-05 0.00000E+00 1.85171E-05 0.00000E+00 2.75114E-05 2.66928E-02 0.00000E+00
```

Where, the header indicates the total number of nodes, the number of columns of information and the time to which the results correspond.

The information in the columns shows the following information:

1. **Number of Node:** the node number of the mesh.
2. **Injection/Pumping:** flow injected (-) or pumped (+) in m^3/sec.
3. **Rainfall:** input water for precipitation or irrigation (-) en m^3/sec.
4. **Real Evaporation:** flow of actual evaporation from free water levels in m^3/sec.

5. **Real transpiration:** flow of actual transpiration from groundwater and soil through continuous (phreatic) or discontinuous (field capacity) withdrawal in m^3/sec.
6. **Diversion:** flow of water diversion in surface water bodies in m^3/sec.
7. **Overland thickness:** surface overland thickness not infiltrated during water recharge in meters.
8. **RAW:** discontinuous water volume remaining in the soil, in meters, that is available to be transpired.
9. **Recharge**: discontinuous vertical water recharge to the aquifer in m^3/sec.

3.1.2.5 Output VEL file

This file stores the velocity fields of the X and Y flow directions of groundwater, subsoil and surface real velocities of water of all the nodes of the mesh.

This is a sample of the information printed in the file VEL:

```
 48274    6   63417600.
     1-6.77267E-06 1.17376E-07-6.20059E-07 1.14821E-08 0.00000E+00 0.00000E+00
     2-8.52730E-06-1.55157E-06-7.88128E-07-1.42883E-07 0.00000E+00 0.00000E+00
     3-9.97980E-06-2.87788E-06-1.00593E-06-2.92474E-07 0.00000E+00 0.00000E+00
     4-1.05937E-05-2.16681E-06-1.06309E-06-3.01636E-07 0.00000E+00 0.00000E+00
     5-1.71060E-05-1.08791E-06-7.29257E-07-3.68541E-08 0.00000E+00 0.00000E+00
     6-2.98085E-05-5.04334E-06-1.24388E-06-2.13727E-07 0.00000E+00 0.00000E+00
     7-3.92611E-05-1.60934E-05-1.56516E-06-6.17067E-07 0.00000E+00 0.00000E+00
     8-2.92795E-05-1.82406E-05-6.92945E-07-4.31576E-07 0.00000E+00 0.00000E+00
     9-2.26753E-05-1.49393E-05-4.60697E-07-2.98928E-07 0.00000E+00 0.00000E+00
```

Where, the header indicates the total number of nodes, the number of information columns and the time that correspond to the results.

The information in the columns shows the following information:
1. **Number of Node:** the node number of the mesh.
2. **Vel.Groundw.X:** component X of groundwater velocity flow in m/sec.
3. **Vel.Groundw.Y:** component Y of groundwater velocity flow in m/sec.
4. **Vel.Subsoil.X:** component X of subsoil velocity flow in m/sec.
5. **Vel.Subsoil.Y:** component Y of subsoil velocity flow in m/sec.
6. **Vel.Surf.X:** component X of surface velocity flow in m/sec.
7. **Vel.Surf.Y:** component Y of surface velocity flow in m/sec.

3.1.2.6 Output VNO file

This file stores the information about fresh/salt water levels and surface water bodies and other variables related with the thickness of fresh/salt waters.

This is the sample with the information printed in file VNO:

```
 48274    5   63158400.
     1 1.95542E+03 0.00000E+00-1.92336E+03 0.00000E+00 0.00000E+00
     2 1.95553E+03 0.00000E+00-1.92556E+03 0.00000E+00 0.00000E+00
     3 1.95566E+03 0.00000E+00-1.92783E+03 0.00000E+00 0.00000E+00
     4 1.95579E+03 0.00000E+00-1.92988E+03 0.00000E+00 0.00000E+00
     5 1.95643E+03 0.00000E+00-1.93159E+03 0.00000E+00 0.00000E+00
     6 1.95842E+03 0.00000E+00-1.93348E+03 0.00000E+00 0.00000E+00
     7 1.96190E+03 0.00000E+00-1.93514E+03 0.00000E+00 0.00000E+00
```

Where, the header indicates the total number of nodes, the number of columns with information and the time that correspond to these results.

The information in the columns shows the following information:
1. **Number of Node:** the node number of the mesh.
2. **Phreatic level:** location of the groundwater phreatic levels and the free surface water levels (meters)
3. **Thickness of groundwater:** thickness of underground freshwater above the impervious substratum (meters).
4. **Thickness of saltwater:** thickness of the saltwater edged above the impervious substratum (meters).
5. **Surface water depth:** free surface water depth/thickness above the soil surface (meters).
6. **Fresh/saltwater interface:** location of the immiscible fresh/saltwater interface, in meters.

3.1.3 Formulations ground/surface models

The model used in water resource modeling, MELEF FSW, is a horizontal two-dimensional (quasi 3D) finite element code for regional superficial subsurface flow of fresh and/or salt water through various types of watershed systems, developed with an implicit (Eulerian) temporally centered (Cranck – Nicholson) and especially centered (Galerkin) numerical approximation. Triangular elements of three nodes allow the analytical integration corresponding to the numerical formulation for the permanent and transient regime. The preconditioned iterative algorithm GMRES provides the solution of the system using a reduced amount of calculation memory and a simple processing of the numerical grid. The recent evolution of these methodologies makes it possible to assess the drainage network of surface runoff and the groundwater levels of fresh and/or salt water, in addition to runoff, sub-alveo or hypodermic flow, recharges and discharges in punctual and diffuse superficial areas, the thickness and velocities of surface and groundwater flow, as well as diversions of surface water in rivers, water balances and the flooding of surface water bodies.

This numerical approach considers novel modelling features of the joint surface and ground water regional flows of continental freshwater and coastal saltwater intrusion from the sea by means of an immiscible fresh/saltwater interface reliable for different kinds of watersheds and environmental problems. Surface and groundwater interactions are depth-averaged through a novel interpretation of a linear river flood routing method. Infiltration rates as well as overland flows generation processes are assessed by a new sub-model which accounts for this kind of surface–groundwater interactions with new evaporation and evapotranspiration processes as a diffuse discharge from surface water, non-saturated subsoil, and groundwater table. Figure 69 is a representation of MELEF-FSW model interaction between GroundWater/SurfaceWater/SaltWater.

Therefore, the present numerical approach couples the cited simulation methodologies of all the water resources of a particular region, or of a river basin, in order to consider the 2D (quasi 3D) flows of freshwater and saltwater for a large variety of hydrological domains, water uses and hydraulic results. The need to manage surface water bodies, both natural and artificial, has required the development of new simulation capabilities. The new solutions implemented in the code now allow simulating dam gates (sluice gate) and drains (pipe-spillways), as well as transient water uses due to well pumping, diversions, and galleries, with automatic returns to the environment (water injections) in any area of the discrete model.

Geographically distributed data are commonly managed by Geographic Information Systems (GIS). In this respect, Python scripts were developed to create the GIS Toolbox for QGIS and ArcMap that are implemented to manage hydrological data, parameters and variables, simulated conditions, and boundaries, as well as output results of the present numerical modeling.

3.1.3.1 Groundwater Model

To properly establish the transient equations in partial derivatives, in two dimensions averaged in depth, which govern the regional continental and coastal aquifers, it is necessary to first define the different phases of flow: freshwater and saltwater. When the so considered immiscible interface is not stationary, Hubbert's hypothesis states that the pressure at a given point is the same when approached from both the freshwater and saltwater phases:

$$(h_f - s)\gamma_f = (h_s - s)\gamma_s$$

Where h_s and h_f are the piezometric heights; γ_s and γ_f are the specific weights of salt water and freshwater respectively; *s* is the position of the immiscible interface.

This can be written in other terms:

$$h_s = \frac{s + Gh_f}{G+1}; G = \frac{\gamma_f}{\gamma_s - \gamma_f}$$

Where *G* is the Ghyben-Herzberg factor that varies between 30 and 42 considering different water temperatures.

Freshwater:

$$n_f \frac{\partial h_f}{\partial t} - n_s \frac{\partial s}{\partial t} = \frac{\partial}{\partial x}\left(K_{xx}(h_f - s)\frac{\partial h_f}{\partial x}\right) + \frac{\partial}{\partial x}\left(K_{xy}(h_f - s)\frac{\partial h_f}{\partial y}\right) + \frac{\partial}{\partial y}\left(K_{yx}(h_f - s)\frac{\partial h_f}{\partial x}\right) + \frac{\partial}{\partial y}\left(K_{yy}(h_f - s)\frac{\partial h_f}{\partial y}\right) + Q$$

Saltwater:

$$n_s \frac{\partial s}{\partial t} = \frac{\partial}{\partial x}\left(K_{xx}(s - p)\frac{\partial}{\partial x}\left(\frac{s + Gh_f}{G+1}\right)\right) + \frac{\partial}{\partial x}\left(K_{xy}(s - p)\frac{\partial}{\partial y}\left(\frac{s + Gh_f}{G+1}\right)\right) + \frac{\partial}{\partial y}\left(K_{yx}(s - p)\frac{\partial}{\partial x}\left(\frac{s + Gh_f}{G+1}\right)\right) + \frac{\partial}{\partial y}\left(K_{yy}(s - p)\frac{\partial}{\partial y}\left(\frac{s + Gh_f}{G+1}\right)\right) + Q$$

Where K_{ij} is the hydraulic conductivity tensor; *p* are the impervious substratum positions; n_f and n_s are the effective porosity of the aquifer accounting for fresh/saltwater movements, and *Q* is the inflow/outflow rate per unit surface.

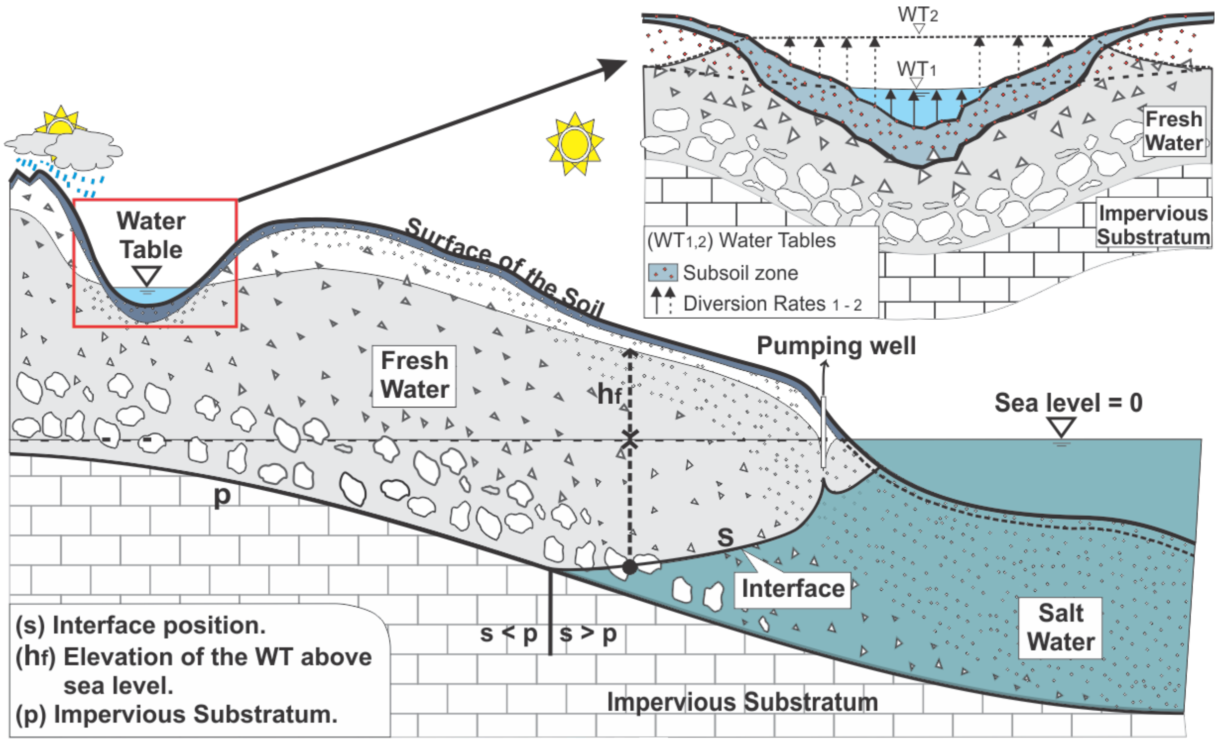


Figure 69: Salt/Surface/Groundwater model interaction of MELEF FSW model.

3.1.3.2 Surface Model

The MELEF-FSW numerical model uses a simplification of the equations governing the free surface flows. In particular, as often happens with other commonly used kinematic and diffusive wave approaches in hydrology, only the classical mass conservation or continuity equation is considered for the present 2D depth averaged model.

Diffusion wave hydrodynamic model

This model comes from the mass conservation equation:

$$\frac{\partial y}{\partial t} + \frac{\partial (V_y)}{\partial x} = \frac{dV}{dx}; \ \frac{\partial y}{\partial t} + V\frac{\partial y}{\partial x} + y\frac{\partial V}{\partial x} = \frac{dV}{dx}$$

And the momentum equation:

$$\frac{\partial V}{\partial t} + V\frac{\partial V}{\partial x} + g\frac{\partial y}{\partial x} + g(I - i) = 0$$

If we make:

$$X = \frac{V}{y}; \frac{\partial V}{\partial y} = -X$$

The momentum equation of the steady diffusive wave:

$$\frac{\partial y}{\partial X} = \frac{i - I}{1 - F^2} = \frac{i - I}{1 - \frac{V \cdot X}{g}}$$

where the bottom slope i, and the friction slope I can be defined as a function of the hydraulic head *h* and the water depth *y*.
Combining both equations, the mass and momentum conservation, we arrive at the diffusion equation:

$$(1 - X)\frac{\partial y}{\partial t} = \frac{dq}{dx}$$

That leads to the transient diffusive gradually varied equation.

Muskingum model

The routing model of Muskingum can be defined as *(I-Q) = dQ + d(I-Q) X*, where *X* is the coefficient of Muskingum. If we make: *d(I-Q) X = (I-Q) dX = (I-Q) X*, we are making the *X* coefficient as variable, and nevertheless *dX* constant and equal to *X*.

Then we can rewrite the Muskingum equation as: *(1-X) (I-Q) = dQ*, o either way: *(1-X) dq = dQ* ; where is made *(I-Q) = dq ; dx dq = dQ = dy dq*, which is a continuity equation similar to the diffusion wave hydrodynamic model, where the Froude number would be equivalent to *X*.

Cattaneo model

Cattaneo's model states that the diffusion equation (in our case, the Darcy equation) is wrong, because it predicts an infinite speed of the water particles (since the solution is asymptotic), which is not possible. The diffusion equation must be written in certain cases, where the diffusion coefficient (in our case the Hydraulic Conductivity, or rather, the Transmissivity) is possibly very high (in the case of surface water) as follows:

$$q = -T^* \frac{\partial h}{\partial x} - Dq$$

which combined to the continuity equation:

$$\frac{\partial h}{\partial t} = -\frac{\partial q}{\partial x}$$

gives the following equation, where τ is the relaxation coefficient:

$$\frac{\partial h}{\partial t} = \frac{\partial}{\partial x}\left(T^* \frac{\partial h}{\partial x}\right) + \frac{\partial}{\partial x}(Dq) = \frac{\partial}{\partial x}\left(T^* \frac{\partial h}{\partial x}\right) + \frac{\partial}{\partial x}\left(\tau \frac{\partial q}{\partial t}\right);$$

for:

$$\frac{\partial q}{\partial t} = \frac{dq}{dt}$$

If we do that $\tau/\delta\, t = X = \alpha \cdot t$ = *constant (where α is the depletion coefficient [T^{-1}])*, we'll have *Dq* = *X· dq*, and the following equation:

$$\frac{\partial h}{\partial t} = \frac{\partial}{\partial x}\left(T^* \frac{\partial h}{\partial x}\right) + X\frac{\partial (dq)}{\partial x};\;\; \frac{\partial h}{\partial t} = \frac{\partial}{\partial x}\left(T^* \frac{\partial h}{\partial x}\right) + X\frac{\partial h}{\partial t};$$

for:

$$\frac{\partial (dq)}{\partial x} = \frac{\partial h}{\partial t} = \frac{\partial y}{\partial t}$$

Where for now we can make the following definitions:

$$X = \frac{\tau}{\partial t} = \frac{\overline{t}}{\partial t} = \frac{dt}{\partial t} = \frac{\overline{x}}{\partial x} = \frac{dx}{\partial x} = \frac{\overline{y}}{\partial x} = \frac{dy}{\partial x} = \frac{\partial h}{\partial x}$$

Where if we define the differentials *dt*, *dx* and *dy* as a function of the hydraulic gradient (∂ *h*/∂ *X*) with respect to the *X* coordinate of the bottom, and making the differential $dy = \overline{y} = y^X$, we can define the water depth variation as:

$$\frac{\partial y}{\partial X} = \frac{\partial \overline{y}}{\partial x} = \frac{dh}{\partial x}\frac{\partial \overline{y}}{dy} = \frac{\partial h}{\partial X}\frac{\partial \overline{y}}{dy} = X' \frac{y^X}{y} = X' \cdot X'' = X$$

where:

$$\frac{\partial \overline{y}}{dy} = \frac{\partial y}{\partial h} = X''$$

With that we have already defined two new factors X' and X''.

The general equation for continuity will be like those found previously from routing Muskingum and diffusion wave models:

$$(1 - X)\frac{\partial y}{\partial t} = \frac{\partial}{\partial x}\left(T^* \frac{\partial h}{\partial x}\right)$$

Differential model

The continuity equation can be written in one dimension in partial derivatives, for the case in which the total differentials are not null:

$$\frac{\partial y}{\partial t} + \frac{\partial q}{\partial x} = \frac{dq}{dx};\;\; \frac{\partial y}{\partial t} + \frac{\partial q}{\partial x} = \frac{dq}{\partial y}\frac{dy}{\partial x};\;\; \frac{\partial y}{\partial t} - \frac{dq}{\partial y}(X) + \frac{\partial q}{\partial x} = 0;\;\; (1 - X)\frac{\partial y}{\partial t} + \frac{\partial q}{\partial x} = 0$$

for

$$\frac{dq}{\partial y} = \frac{\partial y}{\partial t} = \frac{\partial(dq)}{\partial x};\ \frac{\partial h}{\partial x} = \frac{dy}{\partial x} = X;\ Dq = Xdq;\ \frac{\partial y}{\partial X} = X;$$

Where *X* could also be equivalent to the quotient between two expressions of the celerity, one in total derivatives *C* and another in partial derivatives *C' (X = C/C*)*, that is to say:

$$\frac{dq}{dx} = \frac{{}^{dq}/_{dy}}{{}^{\partial x}/_{\partial t}} \frac{\partial y}{\partial t} = X\frac{\partial y}{\partial t}$$

Where:

$$dy = \frac{\partial y}{\partial x}dx + (dx)^2 = \frac{\partial h}{\partial x}\partial x = X\partial x;\ X = 1 - dx;\ dh = dy \approx dx = 1 - X = X\partial x$$

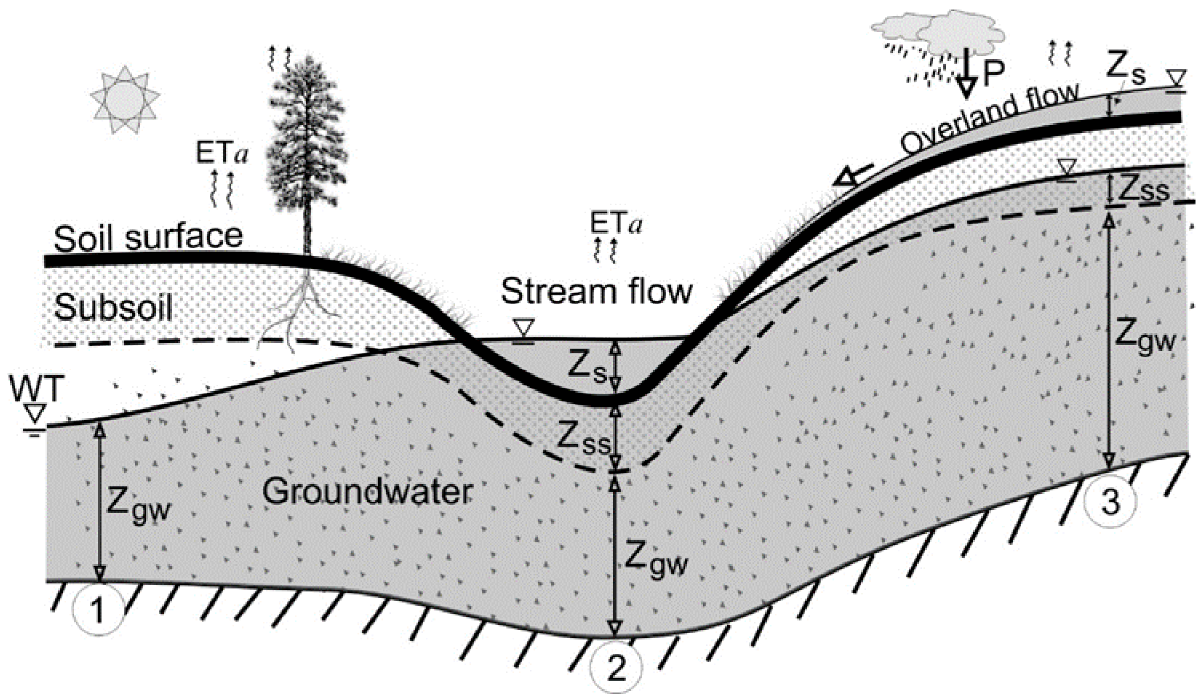


Figure 70: Surface/groundwater flow interaction in MELEF FSW. Thickness of groundwater (Zgw), subsoil (Zss) and stream flow (Zs). Actual EvapoTranspiration (ETa).

Surface model redefined

The surface model can be redefined by considering what follows:

a) Giving a solution to the celerity *C*, is like having a solution to the variation of *q* with respect to the variation of *y* :

$$C = {}^{\partial x}/_{\partial t} = {}^{\partial q}/_{\partial y} = \frac{\partial(dq)}{\partial x} = \frac{\partial(Dq)}{X\partial x} = \frac{\partial(Dq)}{dy};\ \frac{\partial h}{\partial x} = \frac{Dq}{dq} = {}^{\tau}/_{\partial t} \approx X \approx \alpha \cdot t = \frac{dy}{\partial x}$$

$$C = \frac{\partial(Dq)}{dy} = \frac{Dq}{\overline{y}} = \frac{Dq}{y^X} = \frac{Xdq}{y^X} = \frac{\partial(dV \cdot y)}{\partial y} = dV + y\frac{\partial(dV)}{\partial y} = \frac{XdV \cdot y}{y^X}$$

This last can be rewritten in differentials for its integration:

$$\frac{d(dV)}{dy} = \frac{XdV}{y^X} - \frac{dV}{y} = dV\left(\frac{X}{y^X} - \frac{1}{y}\right)$$

Integrating:

$$\int \frac{d(dV)}{dV} = \int \left(\frac{X}{y^X} - \frac{1}{y}\right)dy;\ \ ln\frac{|dV|}{|dV^0|} = X\frac{y^{1-X}}{1-X}\Bigg| - ln|y|;\ \ ln\frac{|dV| \cdot |y|}{|dV^0|} = X\frac{y^{1-X}}{1-X}\Bigg|$$

Which is equivalent to:

$$|dV| \cdot |y| = |dq| = V_0 dx\left(e^{X\frac{y^{1-X}}{1-X}}\right);\ q = V_0(1-X)\left(e^{X\frac{y^{1-X}}{1-X}}\right) = V_0 \cdot f(y, X);$$

q should be a flow per length unity, whilst *f(y, X)* would have unities of length (in this case of artificial thickness). Everything could be expressed in terms of hydraulic conductivity *K*, artificial transmissivity *T** and hydraulic gradient *dh/dx* (where *y* = *h*) in the way:

$$q = V_s(1-X)\left(e^{X\frac{y^{1-X}}{1-X}}\right) = V_s \cdot f(y, X) = -K\frac{dh}{dx} \cdot f(y, X) = -Kh^* \frac{dh}{dx} = -T^* \frac{dh}{dx}$$

Where *X* can be used as being *X* = *X′* · *X″*, and *h** = *f(y, X)* as an artificial water thickness.

b) The function of the celerity *C* keeps the mass for a certain value of *X*.

$$C = \frac{\partial(Dq)}{dy} = \frac{Dq}{\overline{y}} = \frac{Dq}{y^X};\ \frac{\partial(Dq)}{Dq} = \frac{X\partial x}{y^X};\ \ ln(Dq) = \frac{X\overline{x}}{\overline{y}} = X;\ Dq = e^X$$

This also allows establishing that:

$$\frac{\partial(Dq)}{\partial X} = e^X$$

What happens similarly with:

$$\frac{\partial y}{\partial X} = \frac{\partial \overline{y}}{\partial x} = \frac{dh}{\partial x}\frac{\partial \overline{y}}{dy} = X'\frac{y^X}{y} = X' \cdot X'' = X;\ X'' = \frac{y^X}{y};\ \ y = X''^{1/X-1};$$

$$y \xrightarrow{X \approx X'' \approx X' \to 1} e^X \to e^{\alpha \cdot t}$$

This last function would conserve the mass for a determined X close to 1.

c) For the kinematic wave the momentum equation establishes:

$i = I \Rightarrow X \to 1 \to X_{max}$

For the diffusion wave the momentum equation establishes:

$$\frac{\partial y}{\partial X} = i - I = i - \frac{\partial h}{\partial x}$$

or

$$\frac{\partial y}{\partial X} = -\frac{\partial h}{\partial x}$$

if $i \approx 0$ and $X \to 1 \to X_{max}$

If the bottom slope should be considered as significant, the X value could be gradually lower for the terrain *slope* θ.

d) Let us define the X parameter as dependent on the hydraulic gradient for the terrain slope. Let the surface water depth y be the parabola $y(X) : y = AX^2 + B$, where for ($y = 1/2; X = 0$) and ($y = 0; X = 1$) allow us to calculate the constants *A (-1/2)* and *B (1/2)*. The hydraulic gradient has a parabolic behavior, and the following function could then be the appropriate one:

$$2y + X^2 = 1$$

Where the water depth y and the unit coordinate X would depend on the terrain *slope* θ through the following functions:

$$y = {}^{1}\!/_{4}\,(1 - \cos\theta); \quad X = -\cos\left({}^{\theta}\!/_{2}\right)$$

Then, for the horizontal coordinate $x = \cos\theta$ and the groundwater hydraulic head *h*, the so defined hydraulic gradient *X* can be estimated in a hillside of *slope* θ in the following way:

$$X = \frac{\partial h}{\partial x} = -\frac{\partial y}{\partial X} = \frac{{}^{\partial h}\!/_{\partial\theta}}{{}^{\partial x}\!/_{\partial\theta}} = \frac{{}^{\partial h}\!/_{\partial X}}{{}^{\partial x}\!/_{\partial X}} = \frac{{}^{\partial h}\!/_{\partial y}}{{}^{\partial x}\!/_{\partial y}} = -\frac{{}^{\partial y}\!/_{\partial h}}{{}^{\partial X}\!/_{\partial h}} = -\frac{{}^{\partial y}\!/_{\partial\theta}}{{}^{\partial X}\!/_{\partial\theta}} = \frac{{}^{\partial y}\!/_{\partial x}}{{}^{\partial X}\!/_{\partial x}} = -\cos\left({}^{\theta}\!/_{2}\right)$$

That verify also:

$$\cos^2\theta + \sin^2\theta = 1; \ -\frac{\partial y}{\partial h} = \frac{\partial X}{\partial x} = \frac{\partial^2 h}{\partial x^2} =- \frac{1}{4\cos\left({}^{\theta}/_{2}\right)}; \ \frac{\partial X}{\partial \theta} = {}^{1}/_{2}\sin\left({}^{\theta}/_{2}\right);$$

$$\frac{\partial x}{\partial \theta} =- \sin\theta$$

Which give us the interaction of the subterranean hydraulic gradient with the surface water for the slope of the ground surface that can be steep.

Then the steady state gradually varied equation could have also a parabolic solution:

$$\frac{\partial y}{\partial X} = \frac{i - I}{1 - F^2} = \frac{i - I}{1 - \frac{V \cdot X}{g}}; X = \frac{V}{y}$$

That could lead to the transient diffusive gradually varied equation as initially stated.

And that for

$$X =- \cos\left({}^{\theta}/_{2}\right)$$

Which give a solution for the bottom slope:

$$i =- \cos^3\left({}^{\theta}/_{2}\right)$$

This gives us the range of the *X* value, from moderate to high slopes 0.94-0.91 (*i = tan θ*).

Then the parameter *X* can be changed with respect to a considered steep terrain of *slope θ* as it follows:

$$X^* = X - (1 - X)\frac{\partial X}{\partial \theta} = X - (1 - X)\,{}^{1}/_{2}\sin\left({}^{\theta}/_{2}\right)$$

Generally considered moderate slopes do not need any significant change in the *X* parameter. Otherwise, whenever there are steep hill slopes the changes to the *X* parameter should be assessed from the terrain slope.

e) Let us define the flow q' in hydraulics by the conveyance *Kc* (1), and the groundwater flow *q*'' by the Transmisivity *T* (2), where I is the horizontal hydraulic gradient, *y* is the surface water depth and *h* is the saturated water thickness. That is to say:

(1) $q' = Kc \cdot \sqrt{I} = Kc' \cdot y \cdot \sqrt{I}$

(2) $q'' = T \cdot I = K \cdot h \cdot I$

The vertical flows of both must be equivalent:

$$q'_v = Kc \cdot \sqrt{I}^2 = Kc \cdot I = q''_v = T \cdot I^2 = T \cdot I^* \cdot I = T^* \cdot I = Kh^* I = q$$

Which is equivalent to the surface flow q previously defined, where T^*, h^* and I^* would be the artificial transmissivity, water thickness and hydraulic gradient, respectively.

Whence it follows that: $T^* = Kc = T \cdot I^*$

$$I^* = \frac{T^*}{T} = \frac{h^*}{h} = \frac{q''_v}{q''} = \frac{q'_v}{q''} = \frac{q}{q''}$$

The parameter X, for $h^* = f(y, X)$, will respect these relationships when there is surface water, and must conserve the mass for each slope of the terrain.

f) Some equivalences for solutions convergence:
- The value y^X would be in a way the critical hydraulic depth for a section of *slope* θ.
- The parameter *X* would be also the Froude nº, *F* = *V/C*, in this case lower than 1 for subcritical flow.
- This means that *X* depends on the hill slope, as well as on the numerical time step Δt, since *X* is, by Cattaneo, a function of the calculation time step $\Delta t(X = \tau/\Delta t) = \alpha \cdot t$ in transient flow solutions (α is the depletion coefficient).
- The convergence to numerical solution and calibration process must follow the fractions rules described in the section Rules of convergence/calibration:

3.1.4 Reservoir operations

3.1.4.1 Floodgates simulation

The type of floodgates used to control flows in surface water bodies is varied. The gate system supported at two points and whose operation is of the vertical sliding type, is the type of gate that we will call the sluice gate (or vertically sliding).

The sluice gate works based on the following equation in the MELEF FSW code. This equation requires the real opening *h* to estimate the equivalent topography (*h'*) that allows evacuating a given flow based on the internal solution of the surface water depth calculated by the model at each time step. The parameters that control the hydrodynamics of the surface flow in the model will condition the required opening during each time step (simulated opening) to dislodge the same flow as the real opening. The simulated water head (*h'*) will then be reflected in a change in the nodes elevation with a condition of sluice gate that is represented in Figure 71 with the steps 1 and 2, this process will increase or decrease the elevation value of these nodes to allow or close the passage of water.

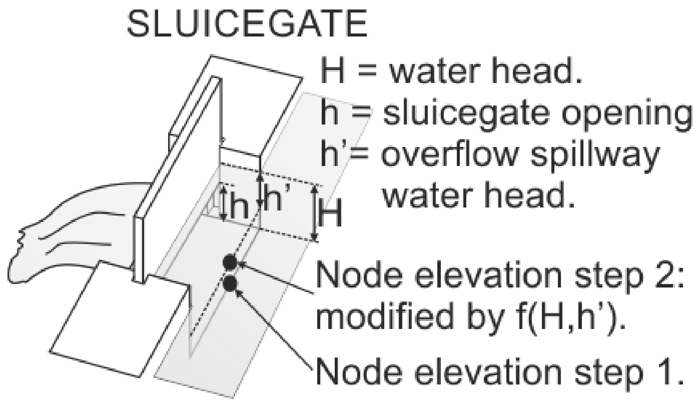


Figure 71: Working scheme of a sluice floodgate

$$h' = 1.31 \cdot h^{2/3} \cdot \left(H - \frac{1}{2}h\right)^{1/3}$$

Figure 71 shows the sluice gate type, where *h* is the actual opening of the gate, and based on this, the elevation of nodes with the sluice gate condition is modified internally in the numerical code and is updated through the following relationship:

$$NodeElev_2 = NodeElev_1 + H - h' \pm h_{\Delta-GATE}$$

Where:

H : Water head of the gate assessed internally.

h' : Equivalent simulated head.

$h_{\Delta\text{-}GATE}$: adjustment parameter of the gate that adds/subtracts to the total opening.

The parameter $h_{\Delta\text{-}GATE}$ serves as height (meters) of adjustment of the simulated opening to increase or decrease the output flow. Its value can be positive or negative, which allows the gate to displace more flow for values less than zero and reduce the flow for values greater than zero.

Then, the new value of $NodeElev_2$ is updated internally to modify the topography and allow the free flow of water, and thereby approximate the flow that would exit if it happened to be a sluice type gate.

Activation of floodgate condition

In the MELEF-FSW model, a series of conditions are required in order to activate as a gate a particular zone, these conditions are:

1. The nodes that participate as a gate zone must have the elevation values of the fields with the following condition *COTA2* > *COTA1* (Spanish names for elevation1 and elevation2 of the nodes) in the attribute table of the GIS layer PRNMELEF. *COTA1* is interpreted as the base elevation or lower point of the gate. *COTA2* can be any value greater than *COTA1*. The nodes elevation *COTA2* can be modified from GIS layer PRNMELEF or from the user interface when PRNMELEF.dbf is processed to create the simulation file PRN.
2. The GIS code of use for the gate should be 9. Equivalently, this code 9 is printed in the simulation file PRN as code 2E9.
3. The model requires the user to generate opening values *h*, in meters, which are the actual gate openings. If it is a single permanent value, you can impose the value in the field *VAL_PERM* of the attribute table of the layer *WaterUses* in GIS (this file is used to generate the SLRMELEF), or with a transient behavior through the Excel file that contains the SLR simulation conditions.

The $h_{\Delta\text{-GATE}}$ parameter can be any value, positive or negative, or even zero.

3.1.4.2 Spillways simulation

The main purpose of intermediate/bottom pipe spillways, in structures such as dams of reservoirs, is to be able to use the total water capacity of the reservoir for management purposes, in addition to evacuating sludge and drag deposits that prolong the reservoir's capacity.

The MELEF-FSW model has been provided with the possibility of simulating intermediate/bottom pipe drains to manage the volume of water in a body of surface water. The implemented solution is limited to pipe drains with a circular cross-section and a longitudinal section without a slope, so that the discharged flow is only due to the hydraulic load on the drain intake point, the diameter of the opening *D* (centimeters) and the concerned losses of water. The solved equation for the pipe drains is as follows:

$$Q(m^3/sec) = D^2 \cdot 1 \times 10^{-4} \cdot \left(\frac{\pi}{4} \cdot \sqrt{2g}\right) \cdot H^{\gamma} \pm Q_{\Delta-DES} = D^2 \cdot f(H) \pm Q_{\Delta-DES}$$

Where:
$D = d \cdot (1 + e^{-0.004d})$: is the diameter of the pipe (cm) adjusted to the practical flow rating curve of the pipe spillway (should be assessed externally to the model itself accordingly to the opening *d*.

$d = d_{EQV} \cdot \beta^{1/4} \cdot C_N^{1/2}$: is the effective opening of the pipe (cm) that would depend on the discharge coefficients, that is, the hydraulic head losses of the pipe system due, for instance, to the installed valve gate itself (sliding, butterfly, etc.), as well as to the pipe inlet, the turbulence, and other frictions in the pipe itself (inlet, bends, lengths, etc.).

d_{EQV} : is the diameter (cm) equivalent to the area of a circular pipe for the each sliding valve shape opening *d'*.

$\beta^{1/4} = (1-\alpha)^{1/4} \rightarrow \alpha$: water head losses in the pipe length.

$C_N^{1/2}$: water head losses at the inlet of the pipe ≈ 0.98

H : is the effective water depth (m); $H = WT - (COTA2 + h_{\Delta\text{-}DES})$

WT : is the water level or the surface phreatic level (m)

COTA2 : is the topographic nodal elevation (m)

$h_{\Delta\text{-}DES}$: is the increment in the drainage level (m)

$\pm\,\Delta Q$: is the maximum change in the drainage

$\pm\,Q_{\Delta-DES}$: is the change in the drainage flow (m^3/sec) = $\pm\,\Delta Q \cdot \left(1 - e^{-0.004H^3}\right)$

$\gamma = 0.5$

The diameter d_{EQV} includes some considerations regarding the equivalent diameter of a circular area, that is, the diameter that equivalently gives the same opening area than the sliding valve shape opening *d'*. Thus, for example, in Figure 72 the valve has an opening *d'* that leaves a particular area for the passage of water. This particular area must be quantified and transformed into an equivalent diameter d_{EQV} (cm) for a circular area. The external sequence of evaluation would be the following $d' \rightarrow d_{EQV} \rightarrow d \rightarrow D$.

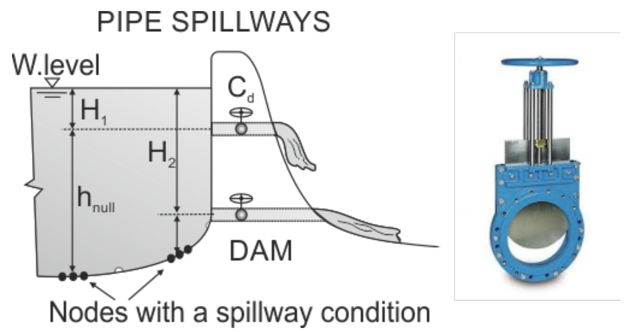


Figure 72: Sketch for small dam with pipe spillways and opening valves.

The type of valves and their closing system is varied, for this reason an equivalent diameter for the same circular section would allow a similar flow rate than the opening of the valve.

Going back to the drain condition, it has two global adjustment parameters:

1. INCREMENT-DRAINAGE-LEVEL, which allows to reduce the topographic elevation of the intake point of all the drains present in the simulation (meters). The purpose of this parameter is not precisely to modify the intake elevation, since this can be resolved from the topography in GIS, but rather the stability of the numerical model. The model with a drain condition can become unstable if the hydraulic head is close to zero or intermittently zero. That is, the sheet of surface water is close to the level of the drainage intake point. To avoid this condition, the drain can be located a couple of meters below the real level and the adjustment parameter can be the difference between the real level and the simulated level (see Figure 72).
2. CHANGE-DRAINAGE-FLOW, which allows to increase or reduce the flow evaluated by the drains (m3/sec). The purpose of this parameter is to adjust the evaluated flow to consider aspects such as pipe losses, numerical leaks (-) from the model or real leaks (+) from the dam itself, and other considerations of flow adjustment.

Figure 72 shows an example where COTA2 is less than the height of the intermediate drain, and the value of $h_{\Delta\text{-}DES}$ will be such that the water depth at the spillway mouth would be increased by the relationship $H = WT - (COTA2 + h_{\Delta\text{-}DES})$.

The management of the drains from the GIS Toolbox is carried out through the code of use 8 in GIS.

Activation of the spillway condition

To activate this simulation condition, the following conditions must be met:

1. The code of use to generate this simulation condition in GIS should be 8, it means that in the attribute table of the WaterUses polygon layer in GIS the field COD_USE should have the value 8. The code of use 8 in GIS is traduced by the user interface to the code 1E9 in the simulation files of the MELEF FSW model.
2. The INCREMENT-DRAINAGE-LEVEL parameter must be different from and greater than zero. Therefore, this parameter, in addition to having the function of increasing the minimum water depth at the drain intake point, is used to enable or disable all the drains in a simulation. To activate the drains, a value other than zero is sufficient, which will normally be positive as described.

Minimum diameter of the drainpipe

The value of the condition in MELEF code must have a diameter greater than 1 (*d* > 1 cm), minimum diameter to simulate a drain. Keep in mind that if the area of a drain is made up of more than one node, the diameter that need to be printed as *D* = *dzone* is the result of distributing the calculable *Q* between all the nodes. In other words, if the area of a drainage is made up of 8 nodes, this zone must have a minimum diameter of about (*D* = *dzone* = *3 cm*), the total flow rate *Q* of drainage must be the sum of 8 drainage nodes with a diameter d = 1.06, which is greater than 1.

$$N^{\circ}_{nodes} \cdot (d^2) = d^2_{ZONE} \Rightarrow d \geq 1 \Rightarrow d = \sqrt{\frac{d^2_{ZONE}}{N^{\circ}_{nodes}}} \Rightarrow D = d_{ZONE} \geq \sqrt{N^{\circ}_{nodes}}$$

Note that the zone diameter minimum value depends on the nodes number of the zone and must be greater than its square root. To avoid possible problems with resulting drain diameters less than d = 1, the user interface rounds any value greater than 0.5 towards 1.001 and values less than 0.5 towards zero.

The drains are also a result of adapting the ability to simulate pumping wells in MELEF FSW to do a double duty. For this reason, the coexistence of the same simulation condition behaving differently requires a borderline. This borderline is the value of the simulation condition, where a value greater than 1 (one) is interpreted as the opening diameter (in centimeters) of a drainage pipe, while values less than one are interpreted as the flow rate in m3/sec of a pumping well.

This frontier, or borderline between both behaviors, disappears when the INCREMENT-DRAINAGE-LEVEL parameter is equal to zero. In this situation, the pumping wells will be able to extract water at rates greater than 1 m3/sec in a single node, anyhow a very huge water withdrawal.

3.1.5 Interaction models

3.1.5.1 Continuous evaporation and transpiration model

The MELEF FSW model considers first there is a continuous evaporation and transpiration process as a diffuse discharge from the water surface or water withdrawal from the water table of the soil unsaturated zone for each node of the horizontal modeled system. The conceptual model of continuous evaporation and transpiration is illustrated in Figure 73.

As can be seen in Figure 73, the actual evapotranspiration *ETa* is evaluated based on the position of the water table with respect to the ground surface. However, when the water table is above the soil surface, evaporation will depend directly on the potential value of evaporation (*EP1, EP2, EP3*, ...) that can be evaluated using empirical functions.

The transpiration curves, from the conceptual model, then start when the water table (*WT*) level decreases in the region located between the soil surface and the soil capillary fringe (*CF*), and their behavior is similar to the empirical relationships that evaluate phreatic evaporation. Therefore, the phreatic evaporation, E_{NF}, is evaluated as:

$$CF < WT < S_s \quad E_{NF} = \frac{EP \cdot b_1}{\left(\frac{S_s - WT}{CF}\right) + b_1} \quad b_1 = min\left(50, \frac{abs(5 \cdot EP - ETP)}{abs(ETP - EP)}\right)$$

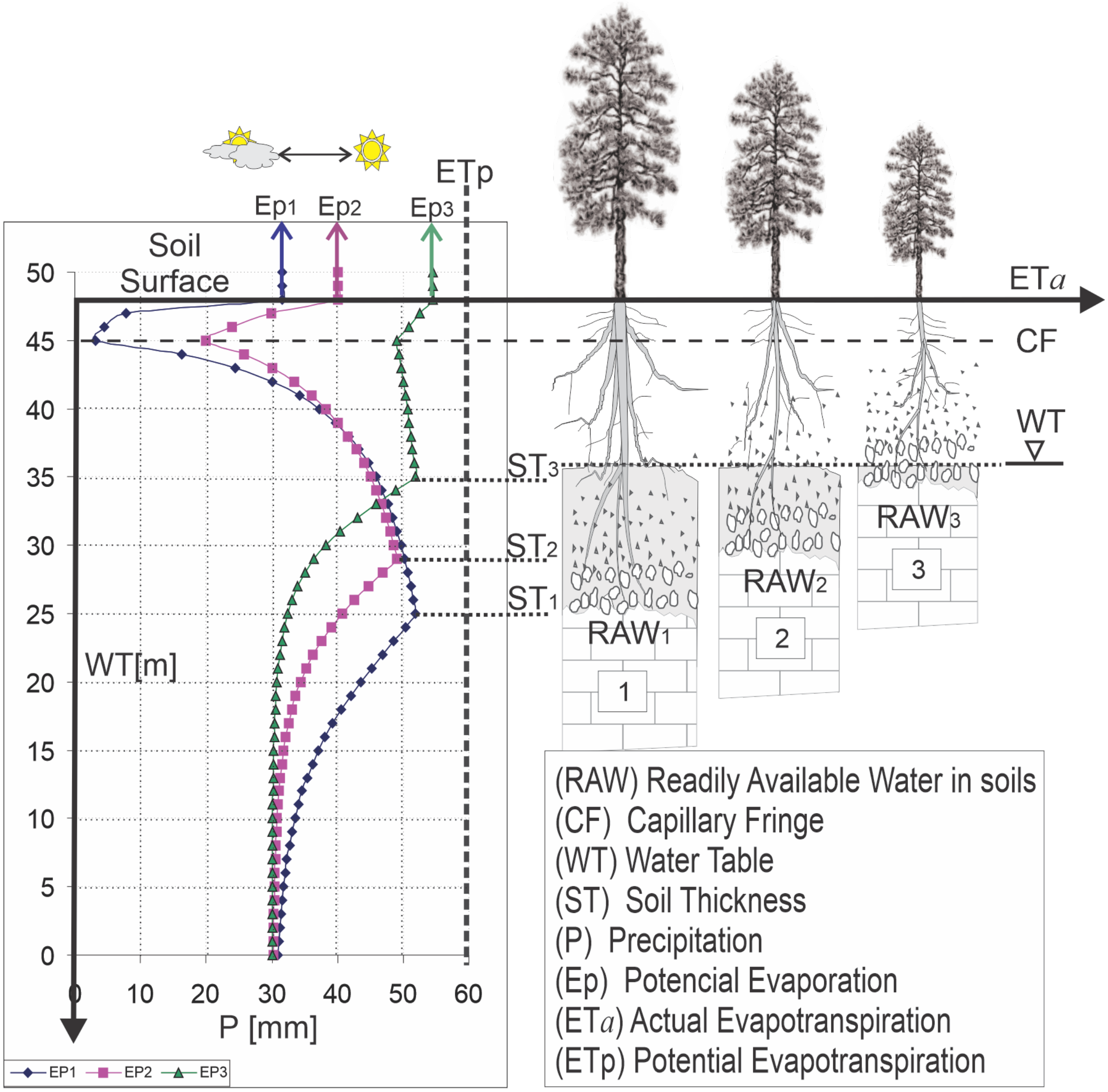


Figure 73: Actual evapotranspiration uptake method interaction.

Where *SS* is the soil surface, *b1* is a coupling function between the previous and the following equations and an adjustment function of the behavior of the *ETa*.

The second part of the transpiration curve begins by increasing towards potential evapotranspiration (*ETP*), in the region delimited by the soil *CF* and the soil thickness (ST), with a behavior like that of other transpiration models. In this region, the transpiration curve is defined by the following relationship:

$$T = ETP - \left(\frac{5 \cdot E_{NF}}{b_1}\right)$$

whenever $ST < WT < CF$

Where *ETP* must be previously evaluated with one of the existing empirical relationships for potential evapotranspiration.

The last part of the transpiration curve begins when the *WT* falls below the depth of the *ST* during the internal evaluation of the numerical code. The transpiration curve then varies gradually towards precipitation (*P*) or towards the *ETP*, that is, if *P* is less than *ETP*, then the curve is limited towards *P*, otherwise, the curve is limited towards the *ETP*. The behavior of the transpiration curve in this area tries to approximate the continuous withdrawal for transpiration of plants under water stress (T_ψ)using the following relationship:

$$T_\psi = P + \frac{(ETP - P) \cdot b_2}{\left(\frac{Ss - WT}{ST}\right)^\alpha + b_2} \quad b_2 = \frac{(ETP - P) \cdot \left(\frac{ST}{CF} + b_1\right)}{5 \cdot EP} - 1$$

Fit parameter α=4; $WT>ST$; if $P>ETP \Rightarrow T_\psi \rightarrow ETP$; if $P<ETP \Rightarrow T_\psi \Rightarrow P$

Where b_2 is a coupling function between the two previous equations, and α is a fit parameter.

This model of continuous water extraction due to evaporation and transpiration from the water table, considers several soils parameters that can have spatial and temporal variability during the iterative process of finding the optimal solution of the numerical system.

3.1.5.2 Discontinuous EvapoTranspiration model

The MELEF-FSW model considers also there is a discontinuous transpiration process as a diffuse discharge from the Readily Available Water (*RAW*, or *RAU* for the spanish acronym) in soils, or water extraction coming from the field capacity.

The conceptual model of discontinuous transpiration, that can be interpreted as the water available for EvapoTranspiration (*ETa*) coming from a percentage of the field capacity, use the well known soil water balance concept to aproximate the *ETa*. *RAW* changes from the beginning (*i*) to the end (*f*) of every time step is explained in the following relations, where to start the evaluation an initial RAW_i is defined considering an initial percentage of water (ini_{factor}):

$$RAW_i^* = RAW_i + P \rightarrow (if \leq RAW_{max})$$

$$RAW_f = RAW_i^* - ETP \rightarrow (if \geq 0)$$
$$ETa = RAW_i^* - RAW_f \rightarrow (if \leq ETP)$$

Discontinuous *ETa* happens whenever there is *RAW* in soil thickness (*RAW* > 0). If *RAW* = 0, only continuous *ETa* could happen. Both, continuous and discontinuous, happen whenever *P* and *RAW* exist during a regular time step, and that in order to better account for every kind of water losses due to evapotranspiration when rainfall happens.

3.1.5.3 Overland flow model

The concept of spatially averaged infiltration capacity, when the soil is saturated (Ks) and the infiltration capacity is considered constant, has been an important assumption for the evaluation of the infiltration rate in the MELEF-FSW code. However, this concept is being reconsidered in hydrology in favor of evaluating the infiltration capacity by applying a Hortonian model that implements a new exponential function on its experimental rainfall-runoff data at a plot scale, which resulted in a better fit than other models that implement it on a constant Ks value. Therefore, the new exponential model is considered to calculate the actual infiltration rate of the soil (I) that depends on the precipitation intensity (*P*).

$$I = I_{max}\left(1 - exp\left(- {}^{P}/_{I_{max}}\right)\right)$$

The equation comes to explain the spatial variation of the infiltration capacity of the soil in a given surface (plot), where *Imax* is the apparent maximum infiltration rate in the plot and *P* is the intensity of precipitation. Therefore, the excess precipitation that generates runoff, or local overland flow, is given by the following relationship:

$$Runoff = P - I = P - I_{max}\left(1 - exp\left(- {}^{P}/_{I_{max}}\right)\right)$$

The equation is evaluated when the water table is below the soil surface, and the excess rainfall, or overland flow, model is then solved.

3.1.5.4 Soil water balance and groundwater net recharge

The conceptual approach of the soil hydrometeorological balance, or soil water balance, allows explaining the process used equivalently by the MELEF FSW model to define the amount of net groundwater recharge and the actual EvapoTranspiration (ETa) in each node of the mesh for each time step. The Figure 74 represents the conditions considered in the soil water balance to find out the RAW_f and Net Recharge values printed as outputs by the model.

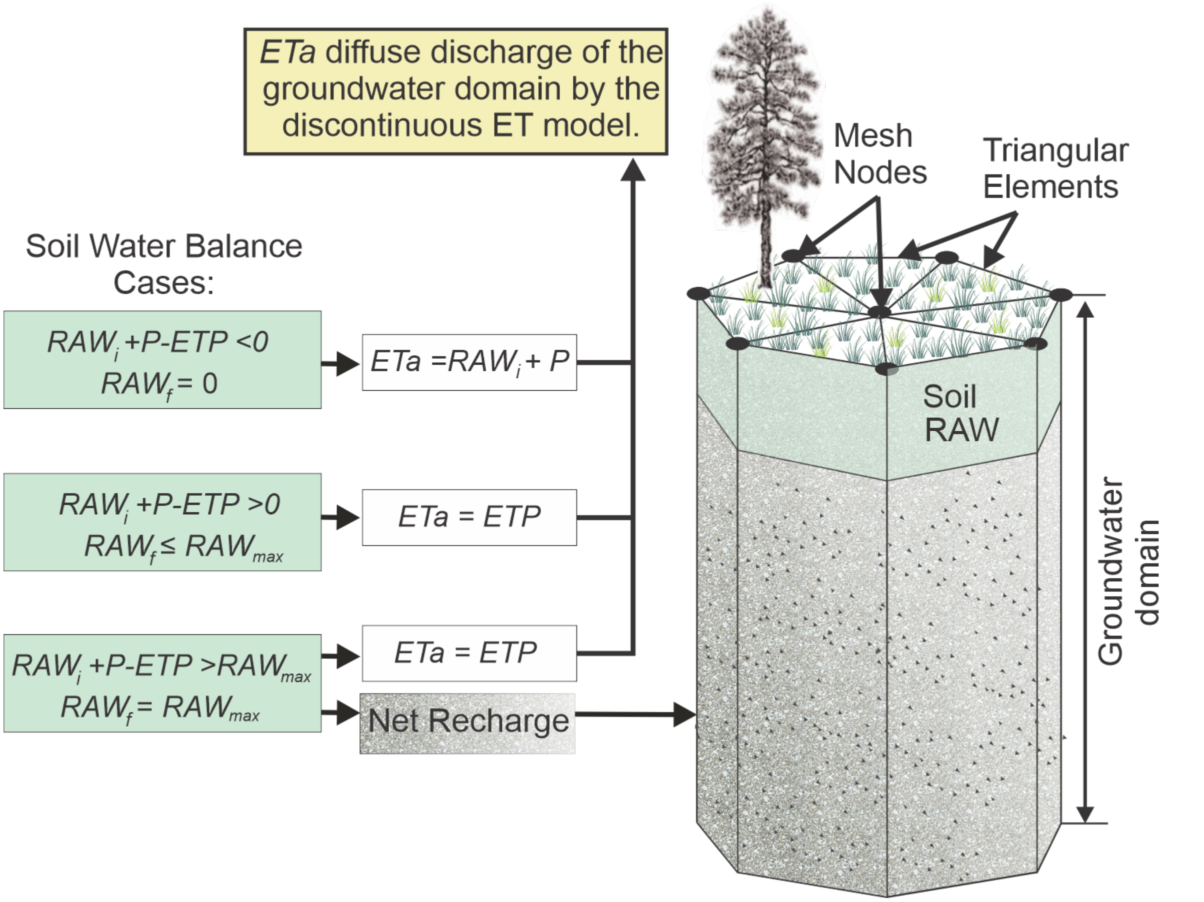


Figure 74: Soil water balance to define disconinuous ETa and the groundwater Net Recharge.

Where in Figure 74 the RAW_{max} is the maximum volume of water retained by soils evaluated as field capacity minus the permanent wilting point.

3.1.5.5 Numerical conditions and resolution

Considering the interaction that exists between the subterranean and surface media, particularly when the water table approaches the soil surface, it becomes necessary to smooth numerically the abrupt change that exists in the properties when passing from the subterranean to the surface media. Based on this numerical need, the use of the hydrological concept of sub-surface or subsoil zone is resumed, which has a role as a layer that serves as an interface of finite thickness with intermediate properties (mainly storage coefficient and transmissivity) suitable for the sub-surface flow in the subsoil zone and serving as a transition zone between both underground and surface media.

Figure 75 shows how transmissivity and effective porosity evolve in their transition from the underground medium to the surface medium. In this interaction, the effective porosity plays a role of groundwater/surface water storage coefficient, which depends on the position of the water table in the groundwater medium, as well as the water depth in the surface medium. The change in the storage coefficient is then gradual between the subterranean, subsoil and surface media. Already in the surface medium, the storage coefficient will always vary towards values close to (*1-X*), which keeps the correct evolution of the surface medium. On the other hand, the transmissivity, which is vertically averaged, gradually varies its value between the subterranean medium and the sub-surface or subsoil zone, as might be expected, although in the surface medium it is artificially increased to achieve a function similar to that of the surface water flow.

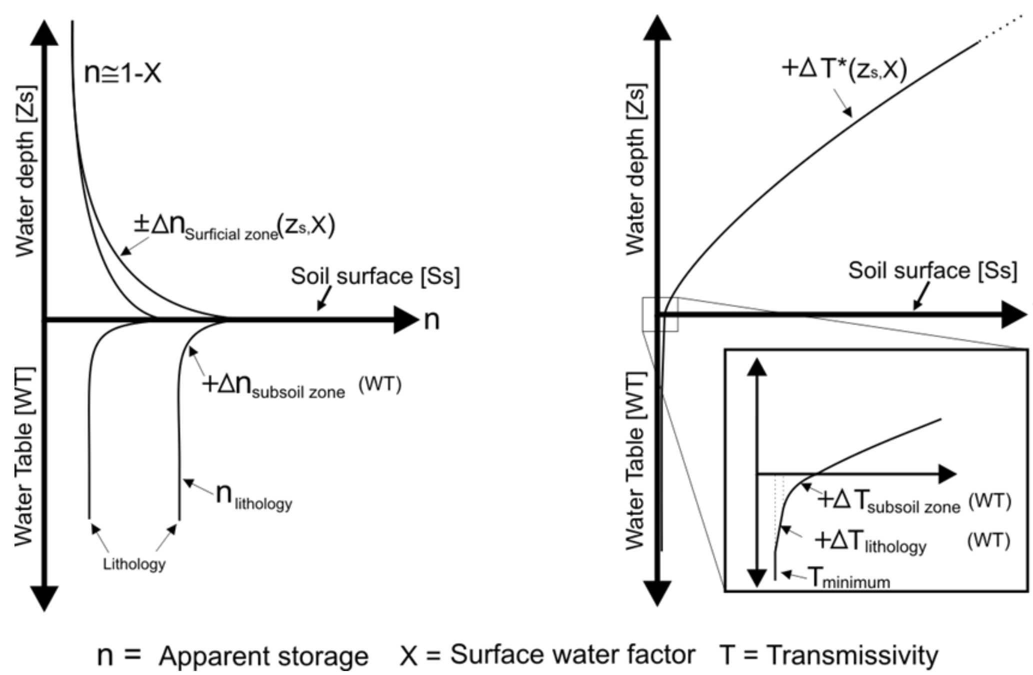


Figure 75: Interaction between underground and surface models, as well as porosity and transmissivity changes.

3.1.6 Tunnels and Galleries simulation

The Infiltration Galleries are horizontally perforated structures with the purpose of intercepting or capturing groundwater to bring it to the surface by gravity. There are various types, although they can be included in a very general way in drains, leaking pipes or more traditionally tunnels. Its constructive design is based on a permeable structure that will be in close interaction with the aquifer, or a geological structure that retains water, while the base of the Gallery or Tunnel may or may not be permeable and will depend on the type and use for which it is intended.

The typologies are diverse, and its operation is particular to the geological structure from which it drains groundwater. Under this situation, the simulation of an infiltration structure, in a 2D model such as MELEF FSW, requires a type of simulation condition that maintains the reality and operation of these structures without excessive parametrization.

The solution to simulate an Infiltration Gallery can be seen in Figure 76, which shows a sketch of a real hydrological condition of a Gallery or Tunnel typology and how it is interpreted in the model. In this, the nodes that represent the path of the gallery are represented in yellow; these nodes have the particularity of having an elevation in *COTA2* less than in *COTA1* as a property.

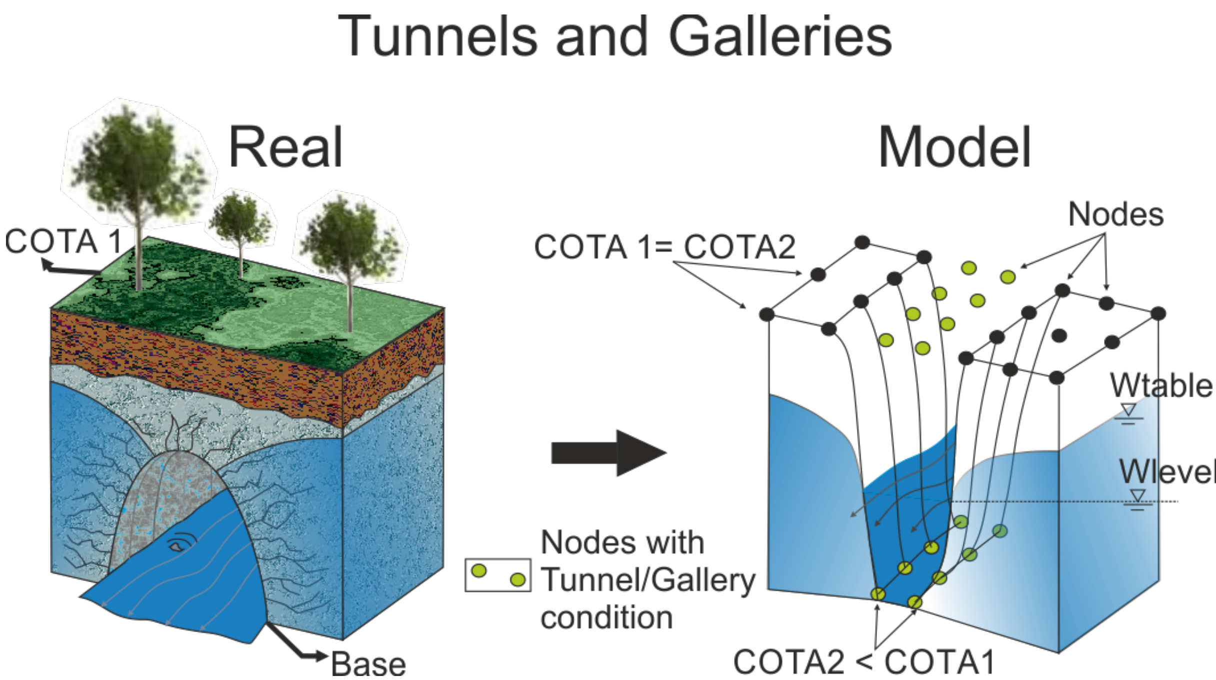


Figure 76: Sketch for tunnels and galleries condition and their activation in GIS

The nodes that meet the Gallery condition, *COTA2 < COTA1*, generate a drop in the topography during the simulation that allows the groundwater to flow as if it were a channel. Likewise, the property of *COTA2* is also the position of the base of the Gallery, and through this, it is possible to establish the elevations of the base of the Gallery along its entire path.

The Infiltration Gallery Coefficient, located in the User Interface, allows establishing a certain control of the simulated collector capacity. This value has a range of (0 – 1), where zero deactivates any infiltration gallery, while a higher value fully activates the

Galleries. At the same time is indicative of the water collector capacity, which will be maximum when the Infiltration Gallery Coefficient is one.

The water collection capacity is modified through the *COTA2* elevation value, when the Infiltration Gallery Coefficient (*CoefGF*) is one, the base of the Gallery is the same as that established in *COTA2*. When the Infiltration Gallery Coefficient is less than 1, the base of the Gallery *COTA2** is modified, depending on the Water Level (*WT*) that exists on the base of the Gallery as follows:

$$COTA2^{*}(meters) = WT - CoefGF(WT - COTA2)$$

By reducing the value of *CoefGF* to values less than 1, but greater than zero, the position of the base of the Gallery is positioned closer to the position of the *WT*, which generates a smaller hydraulic gradient between them.

The interaction between the gallery and the aquifer is resolved internally in the model without requiring new formulations, so other adjustment and calibration parameters are the hydraulic conductivities of the surrounding materials, and the value of the X coefficient that defines the behavior of the surface flow.

Activation of infiltration gallery condition

1. In GIS, modify the field *COTA2* value in the attributes table of the PRNMELEF layer, or also through the DTMsubstratum layer that is required to generate the PRNMELEF layer.
2. In the User Interface, modify the Infiltration Gallery Coefficient to a value greater than zero to activate the Galleries.

NOTE: this Gallery condition does not use a simulation code, so it is possible to define another simulation condition (rain, diversions and water injections, pumping, others...) on the nodes that simulate a Infiltration Gallery.

3.1.7 Automatic water bypass

The correct management of a basin and its water resources implies knowing the different uses of water and its destination, in addition to other highly important implications such as the rational use of the resource, its correct distribution and management. But let us focus on the aspect of knowing where the water extracted by pumping, a surface diversion, a canalization, a supply network system, or any other use that involves the transfer of water from one point to another in the basin.

The ability to simulate different consumptive uses of water in MELEF-FSW remains in the background without the ability to transfer the extracted resource to another environment in real time.

Being able to extract water from the environment, through different conditions of non consumptive use of water, and return it in real time to another point in it, can be considered as a simulation condition of the AUTOMATIC INJECTION/IRRIGATION type, but which can be more correctly defined as a WATER BYPASS.

The GIS MELEF Toolbox is the tool that is responsible for generating the shapefiles and generating the internal structure of their attribute table, as well as processing the different layers to generate the shapefiles, which with the help of the MELEF FSW Interface are transformed into the different simulation files. Therefore, this tool has been modified to consider the management of the new automatic water injection zone coming from the water extraction/drainage zones. To do this, the WaterUses layer (WaterUses.shp) acquires a new column in its attribute table. The new attribute table column has been named “ZONAINYEC”.

Activation of automatic injection/irrigation condition

1. **In the GIS software**: to achieve this AUTOMATIC WATER BYPASS first go to GIS software and start the edition of the WaterUses layer and draw in it a polygon, or select an existing one, with an extraction/drainage condition (pumping well, water diversion or pipe spillway) and fill the ZONAINYEC field with the ZONA number (target polygon in the same WaterUses layer) that receives the automatic injection. All other zones that do not have an automatic injection are left with a value of zero. From here on, the GIS MELEF Toolbox and the user interface take care of the management of the automatic injection zones. NOTE: The ZONA number of a polygon in the WaterUses layer is obtained with the operation FID +1, where FID is a key field that identify each polygon of the layer in the attribute table (after generate SLRMELEF the field ZONA is calculated automatically).
   a. AUTOMATIC INJECTION: This simulation condition implies that the water extracted from the environment is automatically injected in other zone (polygon in the same WaterUSes layer), with a GIS code of use 4 (pumping/injection), injection that is considered recharge to the aquifer, and water losses are only considered through a global efficiency parameter that directly multiplies the volume of water extracted at each time step. This efficiency parameter is found in the "Parameters / resolution" section of the MELEF FSW user interface control panel under the name "INFILTRATION Gallery | EFFICIENCY ByPass". The ByPass Efficiency can use values between the range

[0-1], where 1 implies that there are no water conveyance losses and that 100% of the extracted water is effectively injected into the aquifer.

b. AUTOMATIC IRRIGATION: The activation of this simulation conditions requires that the zone where water is extracted is a pumping well (GIS code of use 4) and the zone (parcel/polygon) to be irrigated should be created in the same WaterUses layer and define a GIS code of use 5 (precipitation/irrigation). The implications of this simulation condition is that the extracted water is irrigated on the parcel increasing the soil moisture content or Readily Available Water (RAW), so the main water losses are by evapotranspiration or percolation to the aquifer if the water retention capacity of the soil is exceeded. NOTE: the polygon or parcel in WaterUses layer can have at the same time a TIMEFILE label to define precipitation, it means that the model can simulate precipitation and irrigation at the same time in the parcel.

2. **In the MELEF FSW user interface**: Prepare the .SLR simulation file using the SLRMELEF.dbf where the water bypass through the WaterUses layer was defined. The field ZONAINYEC identifies the nodes where water will be bypassed from the nodes with an extraction condition, to differentiate between these nodes a negative value is used to define the nodes with an automatic injection and positive value to identify the nodes with the extraction condition (pumping well, water diversion, drainage, pipe spillway). Each negative node precedes the nodes that will inject water into it, in such a way that the nodes with a non-consumptive use condition are distributed in the total number of injection nodes (nodes printed with a negative value). The following shows how the nodes of an area with drainage condition are printed in the .SLR file, and the negative nodes (highlighted in yellow) where the water will be automatically injected.

```
1000000000           1.0000E+02
-5312 11659 11660 13960 -5313 13961 13962 13963 -7202 16229 16230     0     0
0
```

After each negative node (numbers highlighted in yellow color) of automatic injection zone, are printed the nodes with non-consumptive use condition whose flow will be injected. For this example, the nodes [11659, 11660, 13960] inject the extracted water to the node -5312, and so on.

Examples of ByPass activation in GIS:

1. Example of an automatic injection from a Dam Spillway (code of use 8) to an injection zone (code of use 4) in GIS. In Figure 77 the water_uses layer (polygons) shows that the location of Intermediate Spillway (zone 118) is upstream the dam

curtain, if surface water appears in the nodes with this spillway condition then the water is extracted and automatically diverted into the injection zone 117.

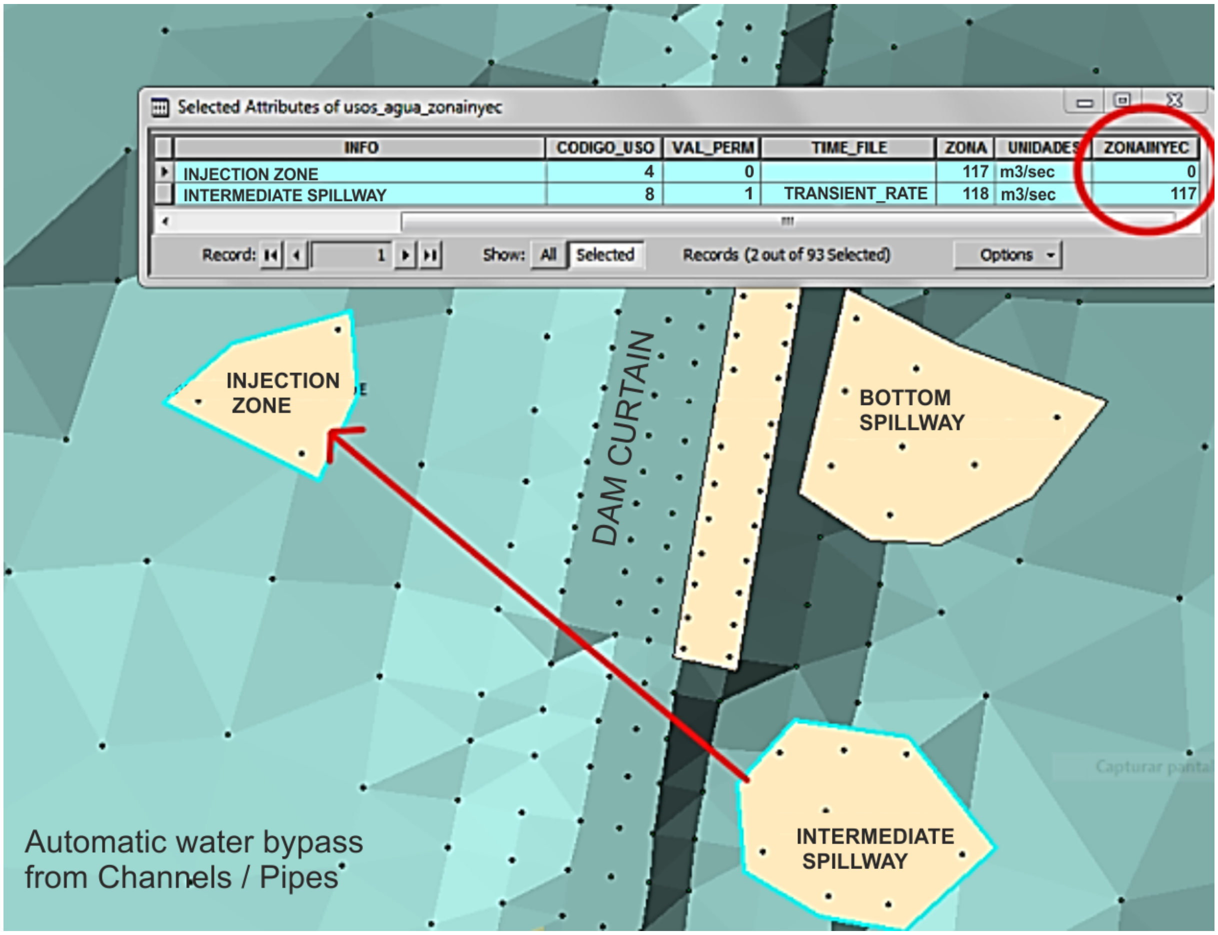


Figure 77: Automatic injection zone managed through the field "ZONAINYEC" in the WaterUses layer of GIS.

2. Example of an automatic irrigation from pumping wells (code of use 4) to a parcel area (code of use 5). In Figure 78 the parcel (zone 32) is automatically irrigated with water extracted from various pumping wells that surround this parcel or are within the parcel boundaries. To enable automatic irrigation it is mandatory to use the code of use 4 to extract ground/surface water, and use the code of use 5 for the parcel where automatic irrigation is to be generated. In addition, the water_uses layer must contain the polygons representing the irrigation parcels in order to manage the "ZONAINYEC" field. In the attribute table the field TIME_FILE for the zone 32 uses the label "C3077", this implies that at the same time there is automatic irrigation in the parcel, there is also rainfall defined by this label through the transient database.

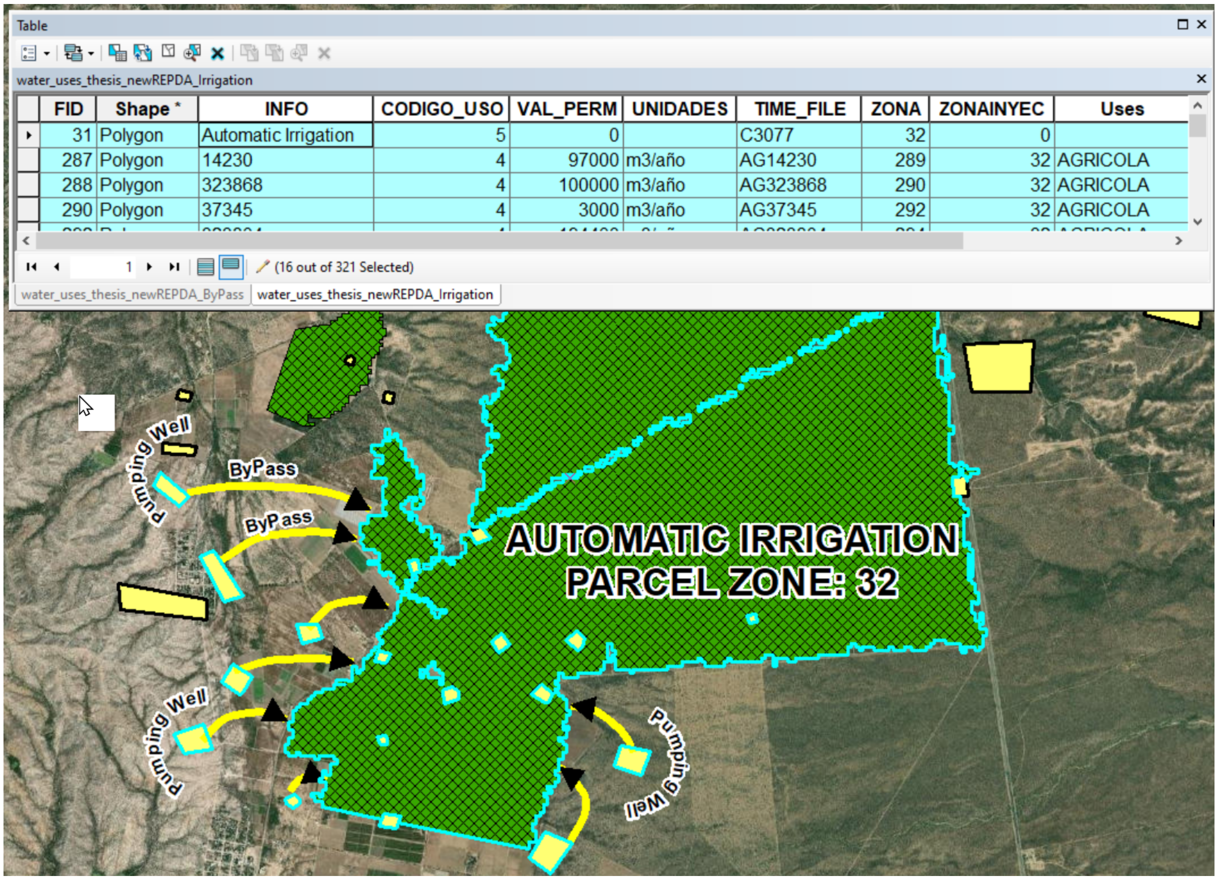


Figure 78: Automatic irrigation zone managed through the field "ZONAINYEC" in the WaterUses layer of GIS.

The new “ZONAINYEC” column then allows you to decide the area where you want to inject/irrigate the water from an area with an extraction/drainage condition. See Figure 77 or Figure 78 to illustrate the modifications made at the attribute table level with the polygon shapefiles that manage the simulation conditions of the .SLR file.

GIS code of use in the automatic injection/irrigation zone.

The automatic irrigation zone must exist with a code of use 5, this due to the structured verification (the interface checks the GIS codes of use to convert them to MELEF FSW codes of use, it verifies that each ZONE is associated with a usage, as well as other checks) that the MELEF FSW Interface performs on the attribute table before writing the .SLR file. In Figure 78 is observed that CODIGO_USO column for the ZONE 32 has a code of use 5, it implies that in this zone there is possible to consider the rainfall of this zone at the same time there is an automatic injection from pumping wells (multiple zones). In case the field CODIGO_USO is left as zero, then the rainfall and irrigation of this zone is not printed in the .SLR file and the automatic irrigation is disabled.

On the other hand, the automatic injection zone (target zone) must exist with a use code 4, for example in Figure 77 zone 118 bypass water to zone 117, and zone 117 has a value in VAL_PERM field of zero while TIME_FILE field is empty. This implies that the target zone can define itself a pumping or injection rate, by changing the permanent value or using a transient database through the TIME_FILE field, and at the same time, receive water from the bypass zones (zone 118). The final result is the sum of the pumping/injection value of the target zone itself and the volume of water transferred from other zones.

Multiple automatic injections and limitations.

As expected, multiple zones with an extraction/drainage condition (diversions, pumping, bottom drainage and others) can use the "ZONAINYEC" to refer to the same automatic injection zone, which means that in these multiple zones the negative nodes are printed to inject automatically the extracted water.

It is important to point out that, there is a limitation in the maximum number of nodes that an automatic injection zone has. This limit of 500 nodes means that if there is an automatic injection into a zone with more than 500 nodes, only the first 500 nodes will receive the full amount of water extracted as automatic injection, and the rest will be maintained with zero injection. This has important implications if the automatic injection is used in large areas.

3.1.8 Rules of convergence/calibration

Use this section to identify the parameter modifications required to fix problems of convergence, numerical oscillations and mass losses in a simulation with MELEF FSW.

In Figure 79, the yellow circles identify the fractions between parameters that can be modified in function of different simulation conditions to improve the model convergence; the circles with numbers identify each fraction of parameters and the order in which these fractions should be planned to be modified starting with the fraction between the length of the mesh nodes (L_{zone}) and the time step (Δt); the circles with letters A and B identify two possible orders of fractions modification.

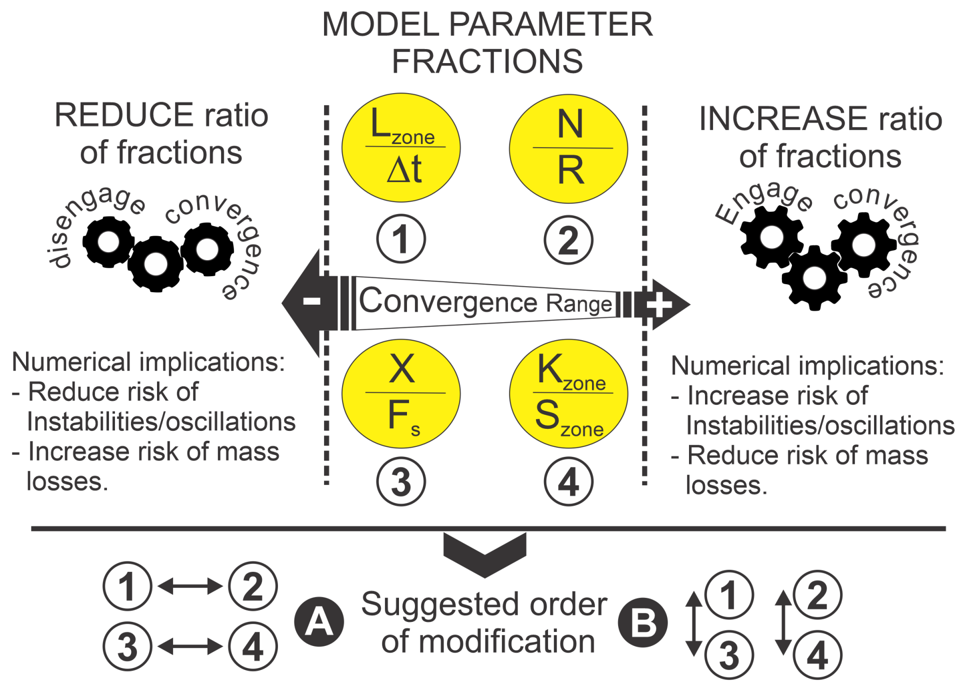


Figure 79: Convergence implications of reduce/increase the ratio of model parameter fractions.

Where:

K_{zone} : Zonal hydraulic conductivity of a geological material.

S_{zone} : Zonal storage coefficient of a geological material.

L_{zone} : Zonal mesh length or distance between nodes.

X : X coefficient (global parameter).

N : size number (global parameter).

R : Iteration Number (global parameter).

F_s : Sub-surface factor (global parameter).

Δt : Time step (global parameter).

The main objective of fractions showed in 79 is to maintain the simulations with MELEF FSW in the range of convergence without numerical instabilities/oscillations. To achieve this you need to consider the following:

1. The distance between the mesh nodes (L_{zone}) should be smaller in the areas where surface water appears, since these areas have interaction of surface water and groundwater and require a higher numerical convergence, but at the same time are a source of numerical instabilities/oscillations if the parameters that maintain influence with the distance between the mesh nodes are not correctly

adapted, these parameters are the time step Δt, the size number N, the number of iterations R and the superficial coefficient X.

2. When the model is running a CMD window appears and the convergence is printed in it as residues NORM, at this window you can check the following:
   a. If printed value of NORM is in a range of 1E-0 to 1E-4: It is a signal that the model is in range of convergence and you can focus on calibration.
   b. If printed value of NORM is frequently greater than 0 or it suddenly crash: It is a signal of numerical instabilities/oscillations, to fix this you can follow the recommendations to modify the ratio of fractions ③ or ④.
   c. If printed NORM is normally lower than 1E-4: It is a signal that the model disengages convergence to some degree and a symptom of this are the the results printed by the model, which look like solutions without or little change over time. To fix this you can follow the recommendations to modify the ratio of fractions ① or ②.

Recommendations to modify the ratio of fractions:

✓ ① $\frac{L_{zone}}{\Delta t}$: The mesh distance between nodes can be modified, especially in zones with surface water and groundwater interaction, or the time step can be modified according to the time resolution of the simulation conditions. NOTE: In some cases is not practical go back to modify the mesh and then you can try to modify the time step first and adapt it to the simulation condition with a higher temporal resolution. For example, if the model simulation use daily rainfall, then a sub-daily time step should be used to engage a better convergence.

✓ ② $\frac{N}{R}$: The global parameters N (NITER) or R (NRDEM) can be modified from the MELEF FSW user interface (these parameters are located in the tab *Parameters/resolution* > *System Resolution*). Considering that increasing these values has important implications in the time required to finish a simulation, it is advisable to try to reduce these parameters as long as the simulation is within the convergence range.

✓ ③ $\frac{X}{F_s}$: The global parameters X and F_s can be modified from the MELEF FSW user interface. Considering that X parameter defines the surface water conveyance and F_s is a factor that defines the transition between groundwater to surface water. These parameters are more important in catchments where the surface water/groundwater interaction is important. If X factor is increased then the surface water conveyance increase (maximum value is 1) and surface water can move faster by the catchment, but so far it has been used for different applications in an interval between 0.92 - 0.94, and try to increase its value beyond this range is theoretically possible by increasing the density of nodes in the mesh and reducing the computation time step mainly. In the other hand is the factor F_s that account for the transition between groundwater and surface water, which allow to increase the aquifer transmissivity before groundwater reaches the ground surface and smooth the difference between the groundwater and surface water velocities. As the factor F_s increases, the subsurface flow also increases (transition zone between groundwater and surface water flow) and this in turn produces an increase in smoothing of the hydrographs and a reduction in surface water flow. Until now, the F_S factor values used in simulations of different catchments with different characteristics are in the range of 100 - 400.

- ✓ ④ $\frac{K_{zone}}{S_{zone}}$: The zonal parameters K_{zone} and S_{zone} can be modified from QGIS/ArcMap modifying the attribute table of the layer Geology. After checking the results, using the 2D graphs of the user interface, it is possible to visually identify the instabilities/oscillations and, with the help of the figure toolbar, to identify the number of nodes involved. In the GIS software search for the number of nodes to find the geologic materials (polygons of the layer) and modify the parameters in its attribute table. Other important implications are: Large differences in the rate of fraction 4 between neighboring geological materials should be avoided, specially when surface water is present; storage coefficient S_{zone} (or equivalent, effective porosity) should be better less than (1-X), specially when surface water is present.

- ✓ If the fractions increase then, gradually the convergence engage and the risk of mass losses reduce but it could produce instabilities (oscillations)

- ✓ If the fractions reduce then, gradually the convergence disengage and increase the risk of mass losses but there should be more stability (reduction of oscillations)

- ✓ A change in the direction of the rate fraction can force a similar change in the direction of the other fractions following the suggested order in Ⓐ or Ⓑ in Figure 79.

3.2 USER INTERFACE

3.2.1 Units of measure

The following units of measurement can be used in the field PERM_VAL of the GIS layers attribute table, or in any database that requires units of measurement other than the base units of the MELEF FSW model.

NOTE: Units in English or Spanish can be written in upper or lower case.

MASUREMENT	BASIC UNITS OF THE MELEF FSW MODEL
TIME	seconds
INTENSITY OF RAINFALL, ETP, EP.	meters per second
WATER VOLUME RATE	cubic meters per second
WATER VELOCITY	meters per second
WATER VOLUME	cubic meters
LENGTH	meters
DIMENSIONLESS	---
FLOW PER AREA AND TIME	cubic meters per square meters per day
OTHER	percentage/100, percentage and degrees

Table 2: Table with basic units of the MELEF FSW

MASUREMENT	UNITS IN SPANISH
TIME	'seg' 'min' 'hrs' 'dia' 'día' 'mes' 'año'
INTENSITY OF RAINFALL, ETP, EP.	'mm/seg' 'mm/s' 'mm/min' 'mm/hrs' 'mm/dia' 'mm/día' 'mm/d' 'mm/mes' 'mm/año' 'mm/a'
WATER VOLUME RATE	'm3/seg' 'm3/s' 'm3/min' 'm3/hrs' 'm3/dia' 'm3/día' 'm3/d' 'm3/mes' 'm3/año' 'm3/a' 'm^3/seg' 'm^3/s' 'm^3/min' 'm^3/hrs' 'm^3/dia' 'm^3/día' 'm^3/d' 'm^3/mes' 'm^3/año' 'm^3/a' 'hm3/seg' 'hm3/s' 'hm3/min' 'hm3/hrs' 'hm3/dia' 'hm3/día' 'hm3/d' 'hm3/mes' 'hm3/año' 'hm3/a' 'hm^3/seg' 'hm^3/s' 'hm^3/min' 'hm^3/hrs' 'hm^3/dia' 'hm^3/día' 'hm^3/d' 'hm^3/mes' 'hm^3/año' 'hm^3/a'

	'L/seg' 'L/s' 'L/min' 'L/hrs' 'L/dia' 'L/día' 'L/d' 'L/mes' 'L/año' 'L/a'
WATER VELOCITY	'm/dia' 'm/día' 'm/d' 'm/hrs' 'm/min' 'm/seg' 'm/s'
WATER VOLUME	'hm3' 'hm^3' 'm^3' 'm3' 'L' 'Litros'
LENGTH	'mm' 'milimetros' 'milímetros' 'cm' 'centimetros' 'centímetros' 'm' 'metros'
DIMENSIONLESS	'adimensional' 'm/m'
FLOW PER AREA AND TIME	'm^3/m^2/día' 'm^3/m^2/dia' 'm^3/m^2/d' 'm3/m2/día' 'm3/m2/dia' 'm3/m2/d'
OTHER	'%' 'grados'

Table 3: Table with units that can be used in Spanish

MEASUREMENT	UNITS IN ENGLISH
TIME	'sec' 'min' 'hrs' 'day' 'd' 'month' 'year'
INTENSITY OF RAINFALL, ETP, EP.	'mm/sec' 'mm/s' 'mm/min' 'mm/hrs' 'mm/day' 'mm/d' 'mm/days' 'mm/month' 'mm/year' 'mm/y'
WATER VOLUME RATE	'm3/sec' 'm3/s' 'm3/min' 'm3/hrs' 'm3/day' 'm3/d' 'm3/days' 'm3/month' 'm3/year' 'm3/y' 'm^3/sec' 'm^3/s' 'm^3/min' 'm^3/hrs' 'm^3/day' 'm^3/d' 'm^3/days' 'm^3/month' 'm^3/year' 'm^3/y' 'hm3/sec' 'hm3/s' 'hm3/min' 'hm3/hrs' 'hm3/day' 'hm3/d' 'hm3/days' 'hm3/month' 'hm3/year' 'hm3/y' 'hm^3/sec' 'hm^3/s' 'hm^3/min' 'hm^3/hrs' 'hm^3/day' 'hm^3/d' 'hm^3/days' 'hm^3/month' 'hm^3/year' 'hm^3/y' 'L/sec' 'L/s' 'L/min' 'L/hrs' 'L/day' 'L/d' 'L/days' 'L/month' 'L/year' 'L/y'
WATER VELOCITY	'm/day' 'm/d' 'm/days' 'm/hrs' 'm/min' 'm/sec' 'm/s'
WATER VOLUME	'hm3' 'hm^3' 'm^3' 'm3' 'L' 'Liters'
LENGTH	'mm' 'milimeter' 'milimeters' 'cm' 'centimeter' 'centimeters' 'm'
DIMENSIONLESS	'dimensionless' 'm/m'
FLOW PER AREA AND TIME	'm^3/m^2/day' 'm^3/m^2/days' 'm^3/m^2/d' 'm3/m2/day' 'm3/m2/days' 'm3/m2/d'
OTHER	'%' 'degrees'

Table 4: Table with units that can be used in English

3.2.2 Excel/CSV standard date formats

Use the following standard date formats to avoid reading errors or loss of information in observed data dates when loading CSV or EXCEL sheets.

STANDARD DATE FORMAT	EXAMPLE OF OUTPUT DATE
'dd-mmm-yyyy HH:MM:SS'	01-Jun-1990 10:30:15
'dd-mmm-yyyy'	01-Jun-1990
'yyyy-mm-dd HH:MM:SS'	1990-06-01 10:30:15
'mm/dd/yy'	06/01/90
'mm/dd/yyyy'	06/01/1990
'dd/mm/yy'	01/06/90
'dd/mm/yyyy'	01/06/1990
'yy/mm/dd'	90/06/01
'yyyy/mm/dd'	1990/06/01
'mmmyy'	Jun90
'mmmyyyy'	Jun1990
'yyyy'	1990

Table 5: Table with standard date formats to use in Excel/CSV databases.

3.2.3 Interpolation as histogram with backward step.

This interpolation method is specially designed to keep the volume under the curve described by the data. This method is defined as an interpolation method as histogram with backward step.

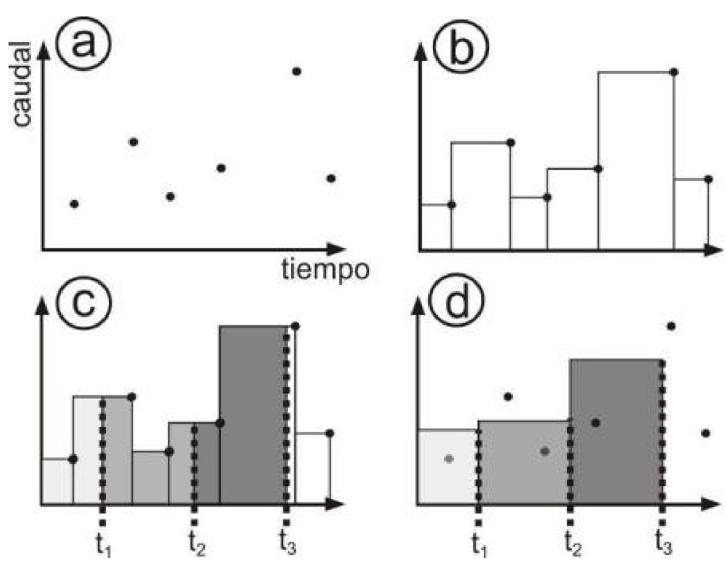


Figure 80: Representation of the interpolation process as a histogram with backward step.

To develop the interpolation method, it was assumed that the hydrological information can only be measured in finite time increments, and in some cases these time increments can be irregular (semi-automatic measurements). Then, the data can be represented in the form of bars, or histograms, where each bar represents the measured value with a thickness equal to its time increment (Figure 80). In this sense, the interpolation divides the bars into the desired times, sums the volume of the bars that are within the new time interval, and finally divides the accumulated volume by its new time interval.

The main properties of this method are to maintain the accumulated volume of the measured data (rates or flow rates), to smooth the data when interpolating with time increments larger than the original ones.
This method is used to interpolate XLS transient databases.

3.2.4 Multiple file selection window

The multiple file selection tool is displayed when any tool, mainly a post-processing tool, cannot automatically locate the files needed to execute an operation. The automatic search criterion implies that the required files have the same name as the file being processed, if this criterion is not met this window is displayed for the user to apply another selection criterion.

The result tools predefine the search path and the different extensions, points *1* and *2* in Figure 81, so that the user only selects the files from the list.

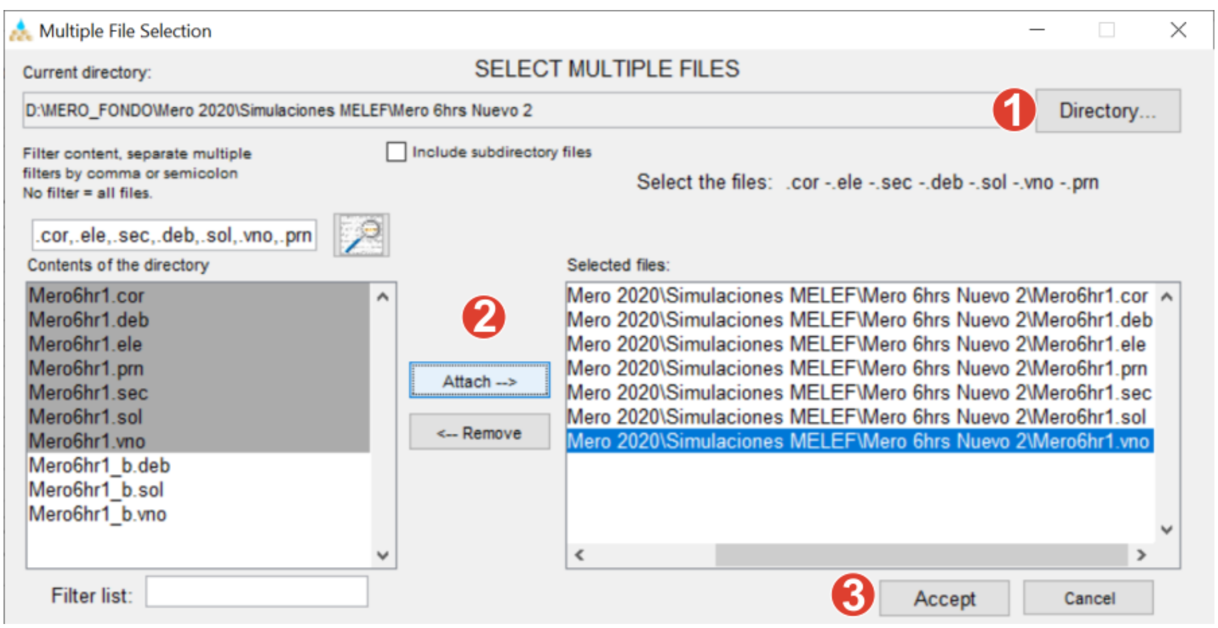


Figure 81: Multiple file selection window.

1: Press the button *Directory...* to select the directory where the result files are located if the automatic search doesn't find the required files.

2: Press the magnifier button to search for the extension files [.SEC, .COR, .ELE, .DEB, .SOL, .VNO] needed. Modify this list if you require another search filter. Check the box -Include subdirectory files- if you want to extend the search to subdirectories. If the search filter is predefined by some post-processing tool, then only select one file for each extension of the search filter, when finished press the *Attach* button to move them to the list of paths confirmed by the user.

3: Press the *Accept* button to send the list of confirmed routes back to the post-processing tool.

3.2.5 Video Recording

Use this tool to configure the video recording properties.

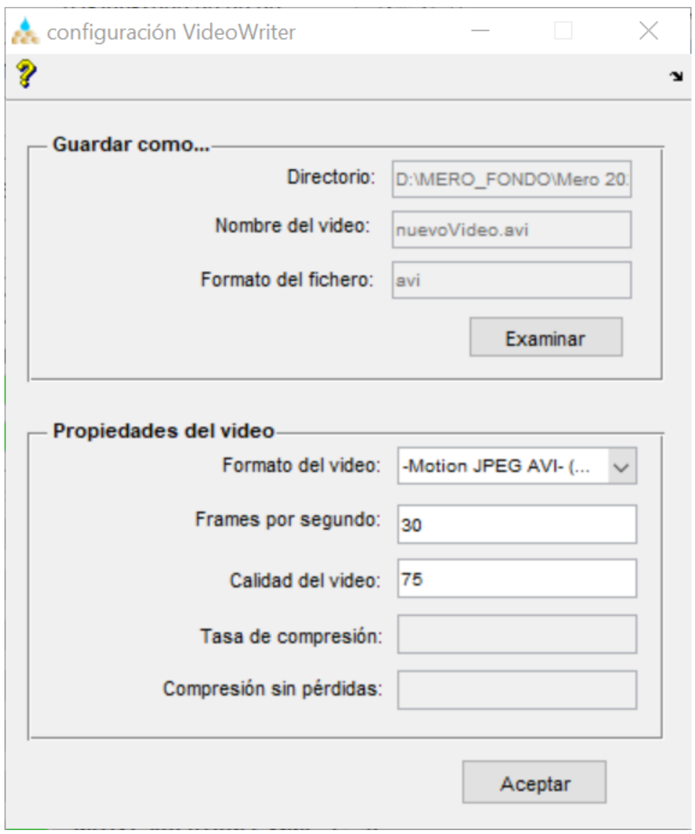


Figure 82: Tool to configurate the video recording settings.

The following is a description of each of the tool properties shown in Figure 82:

Save As...:

- *Directory*: Displays the directory where the video will be saved..

- *Video Name*: Displays the name and extension of the video.
- *File format*: Shows the file format.
- *Examine*: Allows you to modify the directory, video name and file format.

Video Properties:

- *Video format*: the tool can generate five types of videos: [Archival; Motion JPEG AVI; Motion JPEG 2000; MPEG-4; Uncompressed AVI]. The default format is Motion JPEG AVI. The MPEG-4 format requires Windows 7 to be generated.
- *Frames per second*: images per second that will be displayed in the video. The lower the number of frames the video is slower, while the higher the number of frames the video advances faster.
- *Video quality*: when enabled you can define in a range of 0 -100 the video quality, where 100 is the maximum video quality.
- *Compression rate*: when enabled you can set the compression rate of the video. lossless compression: when enabled you can set lossless compression.

If you want to generate a quick video (AVI), then just set the directory and the name of the video and click OK. During video recording do not close or change the size or position of the window being recorded.

3.3 GIS TOOLBOX

3.3.1 Reconditioning of a raster elevation model

The representation of the terrain surface without considering the different objects or natural aspects that exist on it, such as vegetation and different non-natural man-made works, is what we refer to as a terrain or elevation model (DTM or DEM). On the other hand, the digital surface model (DSM) is the one that takes into account all the objects and aspects on the terrain surface.

A digital terrain or elevation model is nothing more than a representation of continuous information through a digital format that can be processed by Geographic Information Systems. These formats are commonly raster and vector.

The raster format basically refers to an image whose pixels represent the elevation information, and the pixel size tells us at what spatial resolution the information is represented. Thus, for example, a raster with a pixel size of 30 x 30 meters will have little information about hydrological details such as a riverbed or structures such as a dam or road layout.

During the development of a DEM, different processes of re-classification of the point cloud and interpolation are solved to obtain a continuous surface. However, there are drawbacks that are not solved in the best way and that are the main source of inaccuracies in the elevation information, this generates the need to recondition the DEM to be hydrologically correct. These problems or errors can be classified as follows:

ERROR TYPE 1: when re-classifying the point cloud generated by the measurement method (it can be by photogrammetry, LiDAR, others...) it is classified as terrain, vegetation, buildings and other aspects that do not belong to the terrain. The points classified as terrain surface are then used to generate the DEM, by eliminating other classifications there are gaps in the point cloud that are resolved by different interpolation methods. Thus, for example, the interpolation process ends up generating smoothed elevations where there should be a riverbed that has vegetation on its banks that hide the morphology of the river. This same interpolation process generates areas with discontinuity in the slope of a river through pools or areas where water tends to stagnate and that do not occur in reality, but that may have their origin in stretches of riverbed with and without vegetation that affect the interpolation process.

ERROR TYPE 2: derives from a re-classification where, despite knowing that they do not belong to the terrain, it is difficult or impossible to establish at which points they should or should not be classified as terrain, so that everything ends up being accepted as part of the terrain. For example, the layout of a road is part of the land surface in most of its layout, but not so in bridges, level crossings, culverts and other infrastructures that allow the crossing of surface water from one side of the road to the other. This category also includes bodies of water such as lakes or reservoirs, where it is decided to maintain the elevation of the sheet of water since the large extensions that they cover make interpolation to determine the bottom of these bodies of water impractical.

ERROR TYPE 3: has to do with the resolution of the raster, since independently of the previous errors, the cell size defines the terrain features that can be observed and can also generate an unpredictable behavior of the elevations as the resolution decreases (larger cell size).

Therefore, it is to be expected that in a DEM the morphology of watercourses and water bodies is generally poorly represented. It is also important to say that there is no procedure to re-establish the morphology of watercourses and water bodies in a

DEM, and if there were, the raster resolution would not allow to represent it correctly.

Figure 83 shows errors type 2, where the circles mark the level crossings or bridges that are represented as if they were dams obstructing the flow of water, and the solid blue line represents the route where the riverbed should be observed.

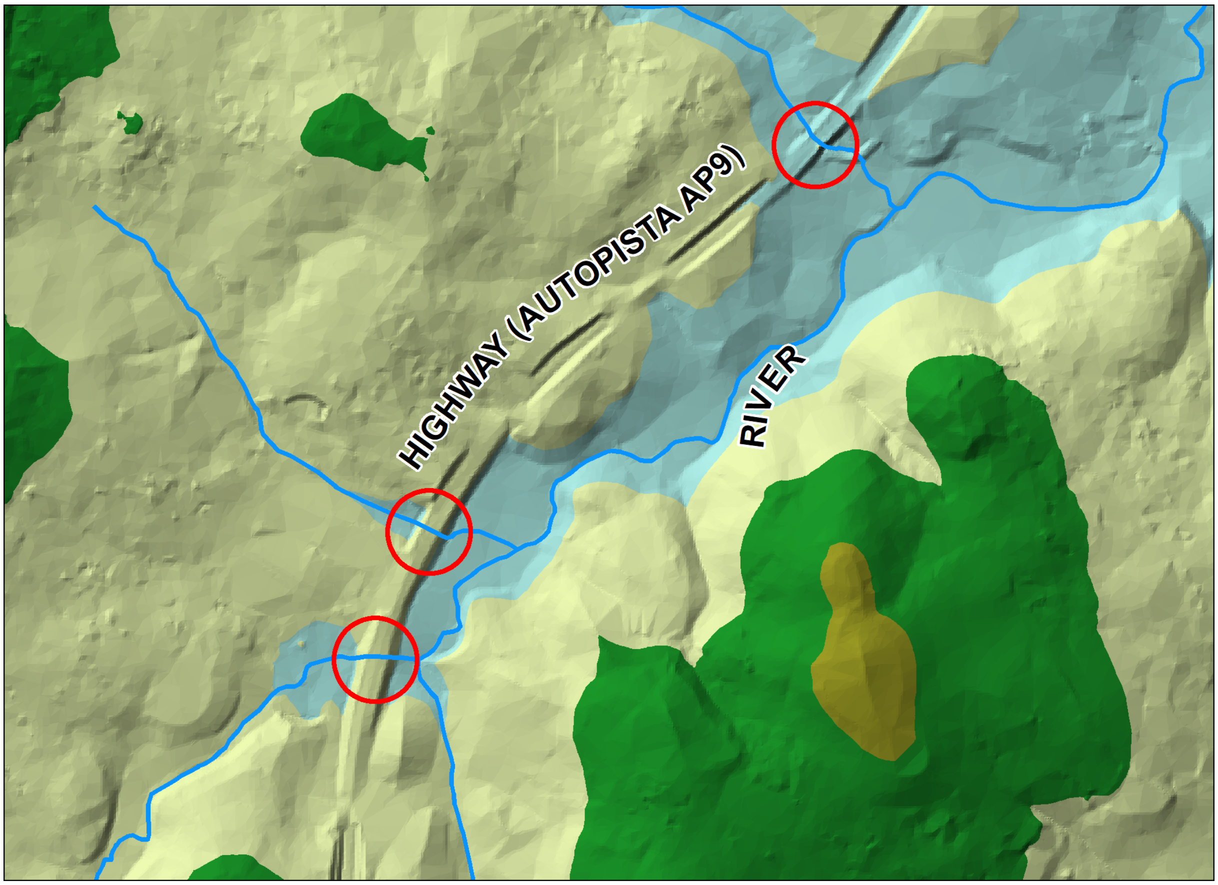


Figure 83: Digital Elevation Model that needs a reconditioning of the terrain elevations to make it hydrologically correct.

A hydrologically incorrect DEM cannot be used for hydrological modeling with MELEF FSW since it would generate false water impoundments, and these in turn would generate changes in the configuration of surface/groundwater levels and flow direction.

Therefore, it becomes necessary to implement methodologies for DEM reconditioning in order to modify elevations and allow surface water flow. It is worth noting again that the reconditioning will be far from reproducing the original morphology of the channels and should not be used as a tool to implement hydraulic models. The only purpose is to create channels through which water can pass in a hydrological model that implements a triangular finite element discretization, and that due to computational limitations this discretization cannot represent in particular detail the morphology of a channel.

Therefore, it is possible to justify the re-conditioning of artificial channels in the DEM (which may have certain similarities with the natural channel in terms of layout, slope, channel width and maximum water depth) to allow hydrologically correct modeling at a watershed scale.

Since it is not possible to generalize with the types of errors that may be present in a DEM, there is no single formula to follow to recondition the raster and make it hydrologically correct. If the following considerations are taken into account the processes to follow may vary for a correct DEM reconditioning:

1. The DEM can present defects of interpolation of the terrain surface and cause areas where water stagnates or stores, and that in reality do not exist, so they must be filled to solve the problem. This type of problem will be contrasted in regions with a steep slope with respect to regions with a very gentle slope or almost no slope.
2. The opposite case also occurs in areas where it is not only possible to fill the land to eliminate topographic depressions where water puddles, since this water puddling area does not exist in reality, but neither is it the result of an interpolation error but has been generated by a barrier represented by a bridge, a level crossing or some other hydraulic work not considered in the generation of the DEM. Therefore, the barrier should be removed and the DEM should not be filled in this area.
3. The channels in the DEM do not always have a downward slope, and this aspect will be closely related to points 1 and 2, where in the case of point 2 it will be necessary to generate a channel that crosses the barrier that stops or dams the surface flow.
4. Finally, it will be necessary to solve the problem of the flow having a channel with an area and slope that are similar to reality, through which the water can flow during the hydrological simulation.

Considerations 1 and 2 can be solved with the tools "Topographic Depression Evaluation" and "Fill Sinks in DEM", point 3 can be addressed with the tool "DEM Burning by Slope" and point 4 can be addressed with the tools "DEM Reconditioning Multiple" and "DEM Burning Elevation". However, it is left to the user's discretion how and in what order these tools are used.

3.3.2 Code of use

This section describes the *code of use* required in the GIS simulation files (SOIMELEF; PRNMELEF; CNDMELEF; SLCMELEF; SLRMELEF) to define the simulation condition, water use or boundary condition you need to simulate. In the attribute table of the GIS simulation file modify the field CODIGO_USO with the code of use shown in the .

GIS SIMULATION FILE	GIS CODE OF USE	MELEF CODE OF USE	DESCRIPTION	MELEF BASIC UNITS
CNDMELEF - CND	1	1	± Continental Imposed fresh water table level (boundary).	meters
CNDMELEF - CND	11	11	± Continental Imposed fresh water table level (boundary). ± Coastal Imposed salt water table level (boundary). NOTE: this code permit to define the fresh and salt water-table level in the same polygon (PERM_VALUE and PERM_VALUE2).	meters
CNDMELEF - CND	2	2	± continental imposed diffuse lateral fresh water flow input/output (boundary).	$m^3/m_{lineal}/sec$
CNDMELEF - CND	21	21	± Continental imposed diffuse lateral fresh water flow input/output (boundary). ± Coastal imposed diffuse lateral salt water flow input/output (boundary).	$m^3/m_{lineal}/sec$
CNDMELEF - CND	3	4	– Continental Open / imposed fresh water table hydraulic gradient discharge (negative value).	m/m
CNDMELEF - CND	31	41	– Continental Open / imposed fresh water table hydraulic gradient discharge (negative value). – Coastal imposed salt water table hydraulic gradient discharge (negative value).	m/m
SLRMELEF - SLR	4	1	+ Punctual pumping – Punctual injection	m^3/sec

			NOTE: the extraction volume rate is divided by the nodes in the zone with this condition.	
SLRMELEF - SLR	5	2	– Recharge; Rainfall; Irrigation (diffuse)	m/sec
SLRMELEF - SLR	6	2	+ Surface water diversion (coefficient)	$m^3/m^2/day$
SLCMELEF - SLC	7	1	– Punctual lateral injection (boundary)	m^3/sec
SLRMELEF - SLR	8	1	+ Spillway openings *d* (cm). The opening *d* in a single node must be grater than 1 (> 1) to activate the Spillway condition, if the zone has more than one node then consider d_{zone} as follows: $d_{zone} > \sqrt{N^{\circ}_{nodes}}; d = \sqrt{d^2_{zone}/N^{\circ}_{nodes}}$	cm
SLRMELEF - SLR	9	2	± Floodgate condition openings (m). To activate this condition it must be defined that COTA2 ~= COTA1 in the GIS PRNMELEF attribute table.	m

Table 6: Table the code of use and condiions required to activate the simulation conditions, water uses and boundary conditions

MELEF FSW HELP MANUAL

Numerical Hydrological Model for
the integrated simulation of regional
groundwater and surfacewate flows
in continental and coastal
watersheds.

Part IV

4 APPLICATION AND TRAINING

To start with this section download and unzip the files of the practical case of the Flooding of Meirama open pit mine (River Mero Basin).

Download: contact with the Editors of this manual.

The exercises presented below are intended to: reproduce the most important steps to generate the GIS layers with the information required by the simulation files, generate the simulation files, run the MELEF FSW model or analyze the model results.

In the application case folder you will find all necessary files to start the exercises in any order, and the project .MAT file with the configuration of all the global parameters, resolution and simulation conditions.

4.1 Practical case 1: Flooding of Meirama open pit mine (River Mero Basin)

4.1.1 Introduction

Since 1980 Meirama coal mine had been exploited by Lignitos de Meirama, S.A. (LIMEISA) as part of an important activity that finished in March 2008. As part of the environmental plan of closure the opencast pit hole was to be flooded to finally create a large lake of ~1.86×10^{6} m^{2} up to a maximum depth level of some 200 m and a capacity of ~146×106 m^{3}. In this context, the forthcoming evolution is analyzed as a function of the actual strategy of flooding. The presently forming pit lake of Meirama is to lie on the drainage basin of the Barcés River, a tributary in the Mero River catchment (247 km^{2}) leading the flow towards the Cecebre Reservoir, north-western Spain, 20 km from the city of La Coruña (Figure 84 and Figure 83).

Figure 84: Flooding of the Meirama open pit (LIMEISA, 2009).

In order to improve the calibration of the stream flow model, continuous water depth measurements were carried out with data logger transducers on 8 of the main streams (Schlumberger Mini and CTD DIVER) at the Mero watershed (labeled with S1 to S9 in Figure 85) from June 2011 until late September 2012. Water depth measurements were corrected by barometric pressure. Water discharge measurements by velocity area and dilution methods, for different regimes, were made on these main streams to obtain rating curves and to transform water depth measurements into water discharges.

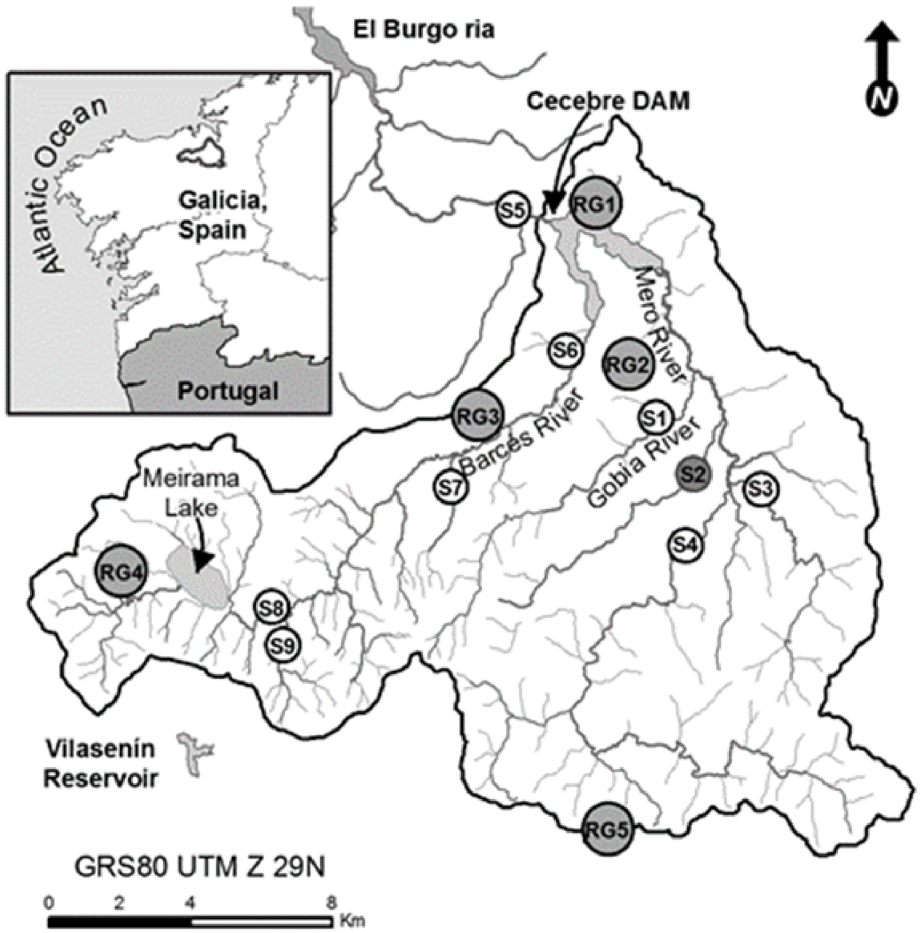


Figure 85: Mero catchment location (~ 247 km2). North-Western Spain. S1-S9 and RG1-RG5 are the locations of measured discharge sections and Rain Gauging of MeteoGalicia Climate Stations, respectively.

With respect to the geological aspects, there are three main regions in the Mero River catchment, one in the NW featured by a fractured and altered granite massif being partially kaolinized, a second one in the S-NE characterized by a schist substratum partially covered by alluvial Quaternary sediments, and a third one marked by a tectonically complex sedimentary basin with Tertiary materials and mining refills which is located between the two former zones of granite and schist and where most of the mining works of the lignite exploitation took place (Figure 86). For these materials, the main hydrologic parameters (hydraulic conductivity, drainage porosity, infiltration rate, soil thickness, capillary fringe,...) have been adjusted during the simulation period (going from January 2008 to September 2012) mainly based, on one hand, on the field measured parameters and on the tectonics and geological features of the region, as well as, on the other hand, on the measurements of the surface water flows, the free surface levels at the Cecebre

Reservoir and at the pit lake of Meirama, as well as on the known groundwater phreatic levels and the water balances inside an area delimited around the pit (Figure 87 A).

The climate in this area is typically Atlantic with an annual rainfall of about 1300 mm and a mean evapotranspiration of 600-800 mm, nevertheless the rainfall in the catchment presents a positive linear gradient conditioned by altitude. To this respect, the catchment might be split up into three different zones by rain gauging station altitudes (Figure 87), where historical daily precipitation rates, temperatures and isolation were considered during almost five years 2008-2012. Thereafter, daily historical precipitation was used and potential evapotranspiration rates were estimated with climate variables over the drainage basin.

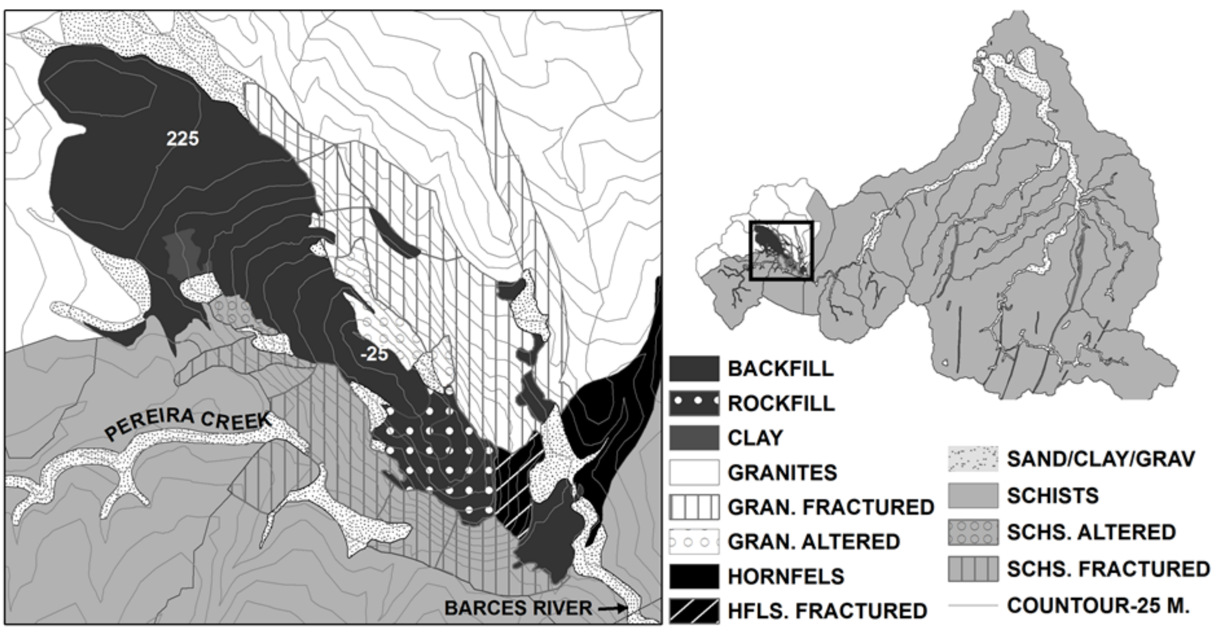


Figure 86: Geology of the Mero catchment area: zoom in the Meirama open pit mine zone

During the last exploitation period (until March 2008) some twenty pumping wells were placed at Meirama pit mine contours to drawdown the water table. The water pumped from the bottom of the pit was then led to a treatment plant, being afterward drained by means of two perimeter channels towards the Barcés River, tributary of the Mero River (Figure 87 A). In order to prevent or to allow the surface water getting into the pit during the flooding period (after March 2008), the inlet streams were gradually diverted through adjustments of flow rates in weirs (Figure 87 A). Other water uses within the Mero River catchment area of main interest are, for instance, the registries of the regulation operations provided by the water supply company of La Coruña, EMALCSA (mainly the flow rating curves and the openings of spillways and floodgates) for the surface water outflows at the dam outlet of the Cecebre Reservoir (Figure 88), as well as others surface water diversions for water supply for industries and neighboring municipalities.

With respect to this, the present work applies the MELEF model to the freshwater resources of the whole Mero River catchment, which includes, among others, the regulation operations of a free surface reservoir and the exploitation, closure, and hydraulic restoration of the opencast mine, with the aid of the historically registered hydrological parameters and data. Thus, the simulation strategy considers a period of almost 5 years, going from 1-January-2008 to 30-September-2012. The simulated conditions consider measured daily rates of precipitation, and the main management of groundwater and surface water at the Cecebre Reservoir and the Meirama coal exploitation and flooding of the mine. Once the flooding was initiated on 18-March-2008, most of the existing wells stopped pumping and the flooding of the mine was started with the groundwater flow. After 3-October-2008 the mine is being flooded with water diverted from some of its neighboring streams (see Figure 87 A). After 15-June-2011, the surface flow of the main stream (Pereira creek), which up to that date had been diverted to the open pit, was thereafter split up, by means of a weir constituted by a spillway (with a fixed opening) and an overflow system (with fixed position), into both the flooding of the Meirama lake and an attempt to reach the minimum environmental flow which was to be supplied downstream through a tunnel to the Barcés River. Thereon, the mean monthly value of the ecological discharge was estimated as about 0.2 m3/s.

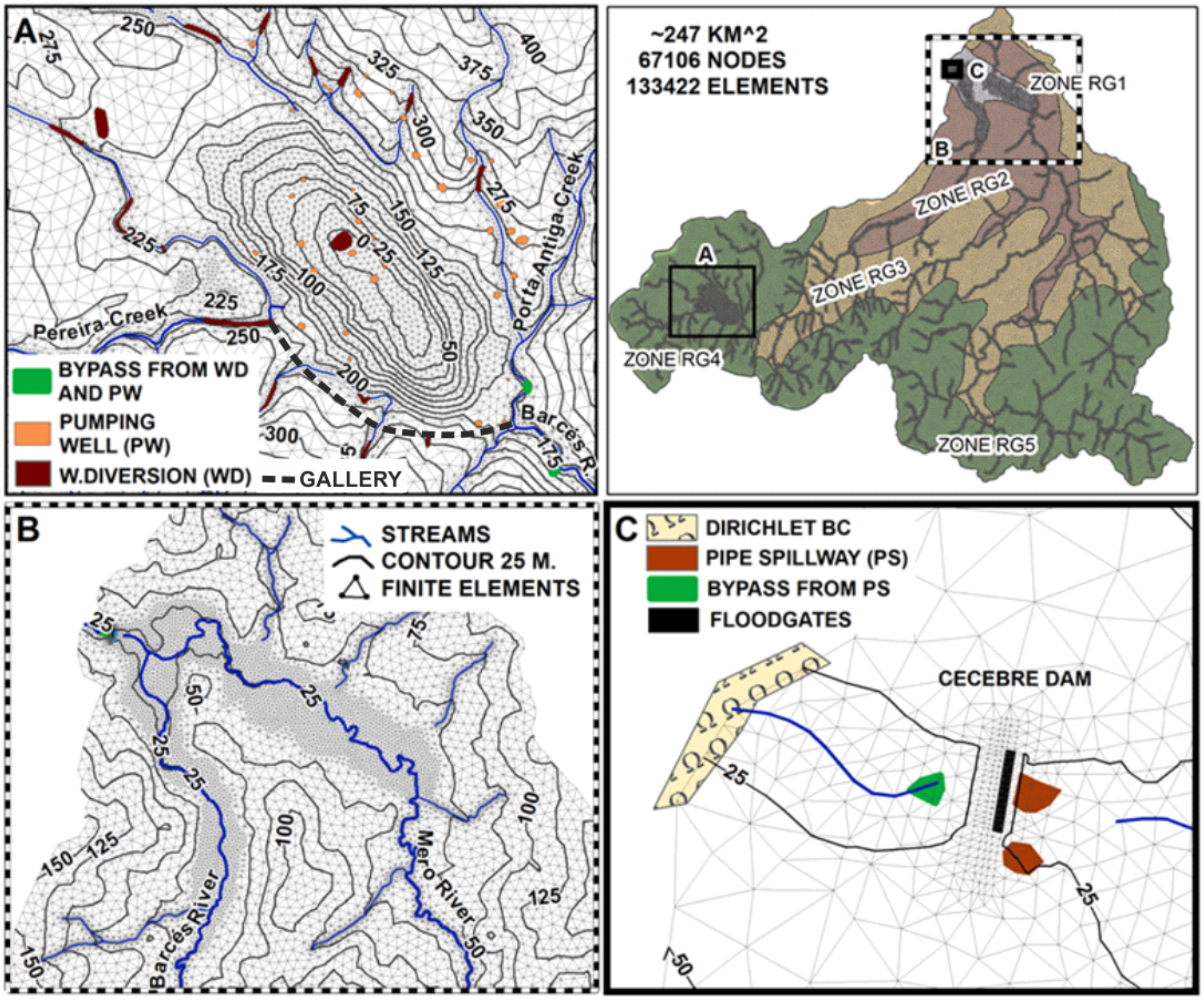


Figure 87: Finite elements mesh model and rainfall zones in the Mero River catchment area. Water management and boundary conditions at the Meirama open pit (A) and the Cecebre Reservoir (B, C).

Otherwise, it is quite a challenge to reproduce the outflow regulation operations of Cecebre Reservoir dam at its outlet, through the particular and changing openings of its five floodgates and its three submerged pipe spillways (one near the middle, and two deep), see Figure 88 C. In respect of this, the flow rating curves of the pipe spillways were calibrated, on one hand, to the actual flow rating curves of the middle pipe, and on the other hand, to the two deeper pipes being considered together. Therefore, the diameters of the pipes were calibrated for the different openings of the valves installed at the pipes and the corresponding discharge coefficients, and this according to the flow rating curves provided by the operator (Figure 88).

With respect to the five sluice gates (vertical lift floodgates), they were placed together in the numerical model, as if they were just one bigger floodgate (with lumped position, shape and openings), during the almost five years of operation, and that in order to try to reproduce their hydraulic behavior from the point of view of the surface discharges as well as the surface and groundwater levels in the reservoir and its surrounding area (Figure 87 C).

The operating policy of this quite small Cecebre Reservoir requires that there should be a safeguard to ensure the filling up to about 70 % of its total capacity, i.e. 14.5 $\times 10^6$ m^3, from November to March, both of them inclusive to prevent eventual floods. From March on, the management is focused on filling the reservoir as much as possible, about 20.5$\times 10^6$ m^3, in order to meet the demands that take place during the summer.

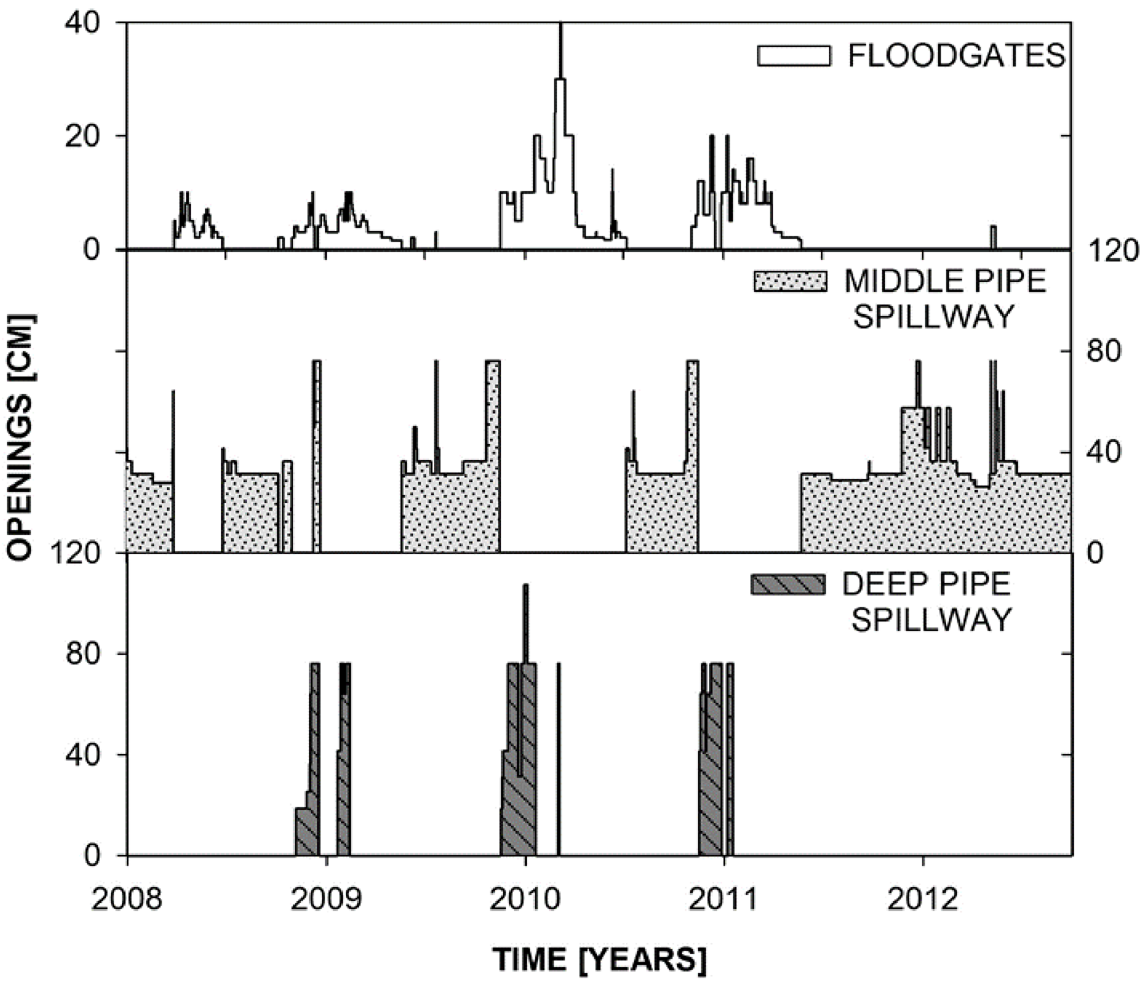


Figure 88: Water management in the outlet of the dam of the Cecebre Reservoir. Openings of pipe spillways and floodgates.

4.1.2 Objectives

The practical application of the MELEF-FSW code to the Mero River basin, with special emphasis on the Meirama mine, had the following main objectives:

- Calibrate the hydrology of groundwater and surface water over a well-known period.
- Simulate the future evolution of the regional water regime until the level of discharge of the Meirama mine pit to the Barcés River is reached.
- Analyze the interaction of groundwater and surfacewater flows in the Meirama mine and the Cecebre reservoir areas during the simulation period using water levels and hydrological balances.

4.1.3 Practice 1: Preparing the numerical model:

Objective of the practice:

- o configure the simulation duration, as well as the parameters and resolution of the system, for the numerical model of practical case of the flooding of Meirama open pit mine, with the help of the model's user interface.

4.1.3.1 Preparing the Simulation conditions

Follow the steps and figures to configure the the panel *Simulation conditions* of the project.

1. **Open the User Interface MELEF FSW and open the project .MAT inside the folder *Mero 2020, see Figure 89.***

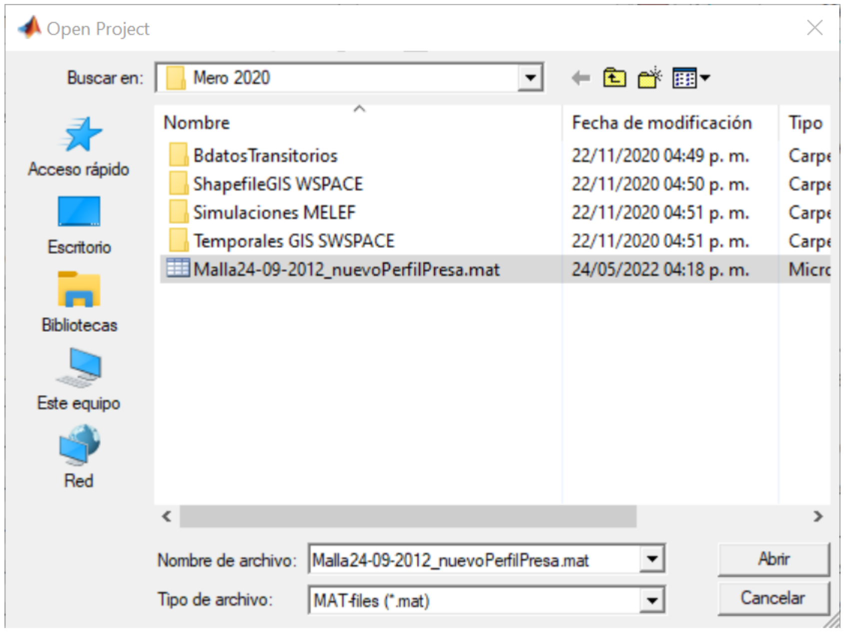


Figure 89: Open the project of Mero River Basin with the practical case of the flooding of the Meirama open pit.

2. **Go to the panel *Simulations conditions* and define the simulation files if the status box is black (defined = green, not defined = black), see Figure 90.**
   - ***a.*** Click on the button *Define ...* of each simulation file and take note of the file name highlighted with a red box.
   - ***b.*** Select the simulation files from the folder *Mero 2020> simulaciones MELEF*.
   - ***c.*** The status box will change to green color after define the simulations file except the simulation condition SLC which is not used.
   - ***d.*** Save changes

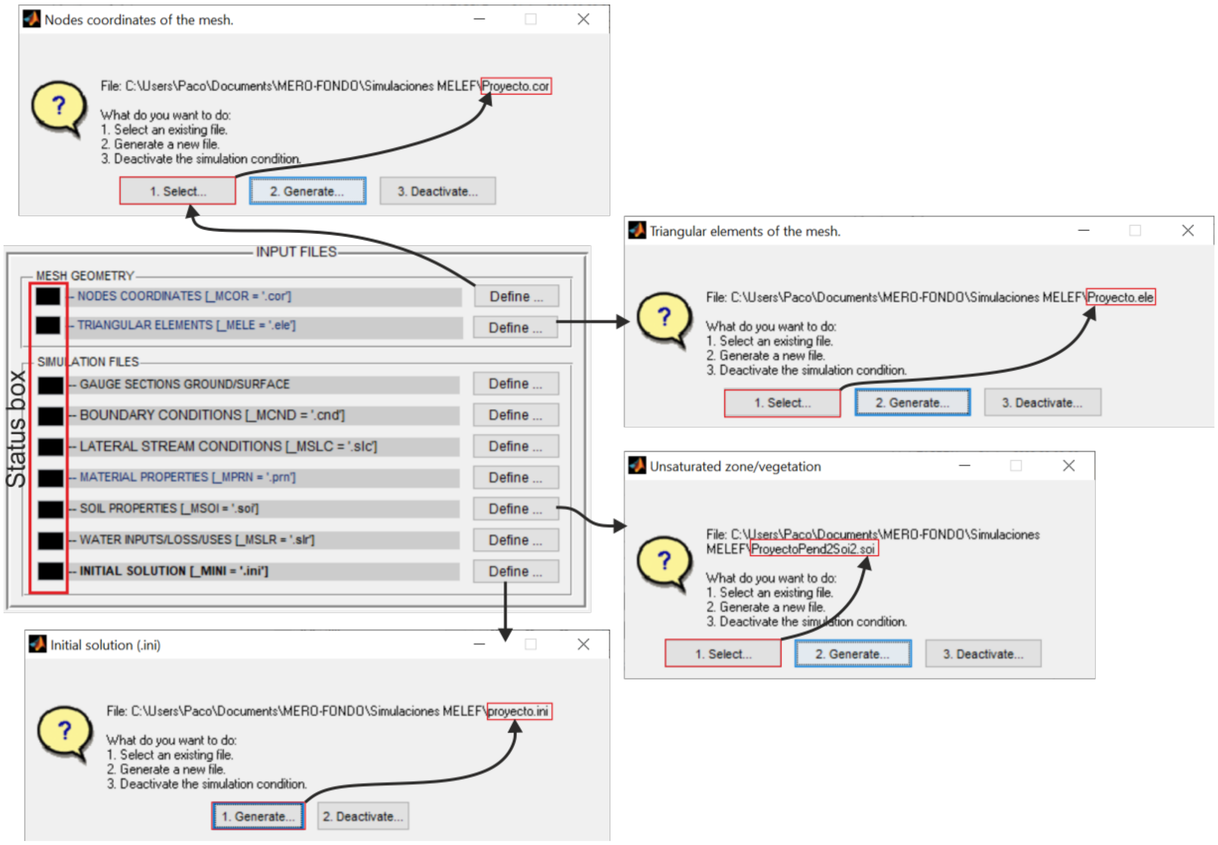


Figure 90: Process to define the simulation files if the status box is black.

3. Defining the initial solution with the following steps of the Figure 91:

1: after click the button *1. Generate ...* the window to define the initial solution appears. In this window click on the button *Examine* to select an initial solution.

2: Select the file *proyecto.ini* and clic *Abrir* or *Open* button.

3: Select the initial solution to use, the number 450 is the simulation day in which the groundwater level solution for each node of the mesh was printed in a previous simulation.

4: Press the button *Apply configuration* and check that the color of the status box change to green.

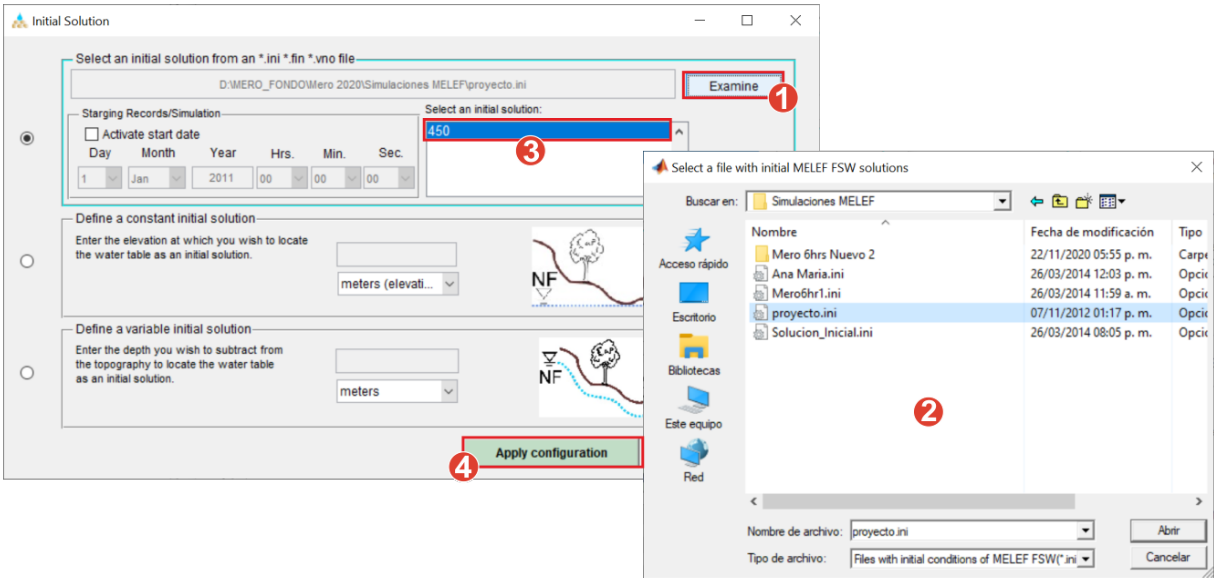


Figure 91: Process to define an existen initial solution file.

4. **Configuration of the *Time Control* to set the duration and calculation time steps (Figure 92):**
   - ***a.*** STARTING RECORDS/SIMULATION: 1-jun-2006
   - ***b.*** TIME STEPS: 6 hours
   - ***c.*** END DATE OF SIMULATION: 30-sep-2012
   - ***d.*** NUMBER OF TIME STEPS: After define the start and end dates of simulation and the time step, the user interface fills automatically the number of time steps required to finish the simulation (9522).
   - ***e.*** READING SETTINGS: check the box *Activate reading leap (_TINI)* and define as a new starting simulation date 01-jan-2008, it implies that the model doesn't need to start the simulation since 1-jun-2006 and apply a reading leap to the date 01-jan-2008 in all simulation files. This option can be useful in different situations, perhaps to restart a simulation after a power failure or simply for calibration purposes in a specific period of the simulation, among other possibilities.

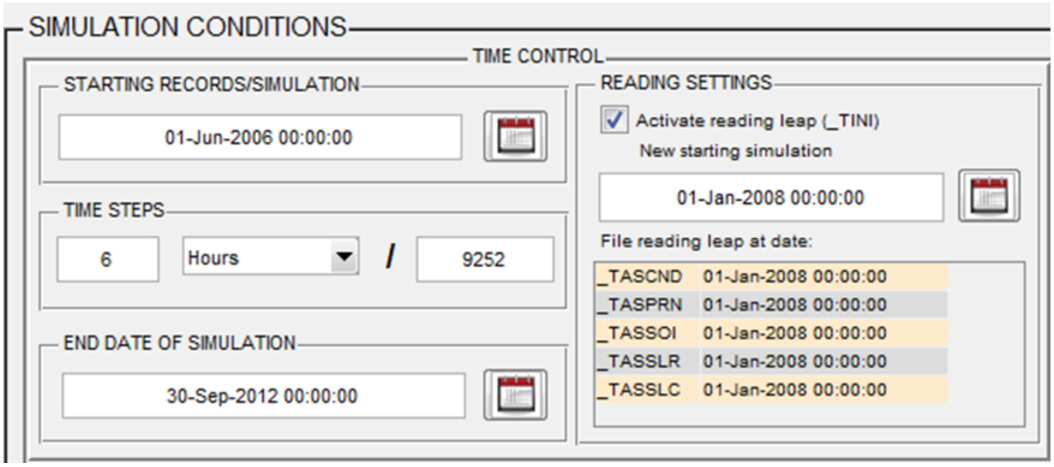


Figure 92: Configuration of the Time Control for the project Malla21-09-2012_nuevoPerfilPresa

5. **Define the name of the simulation files once these are copied to the execution folder execution (Figure 93).**
   a. Define the name: Mero6hr1 or other name with a length equal or less to 8 characters.

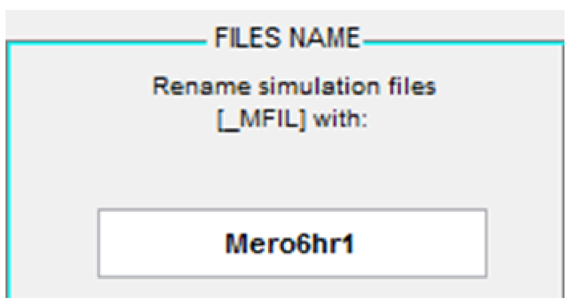


Figure 93: Define the name of the simulation files once these are copied to the execution folder.

6. **Configuration of *Output files* or model results (Figure 94).**
   a. Define which simulation results you want to print, as well as the printing frequency every number of time steps, see the proposed configuration in Figure 94.

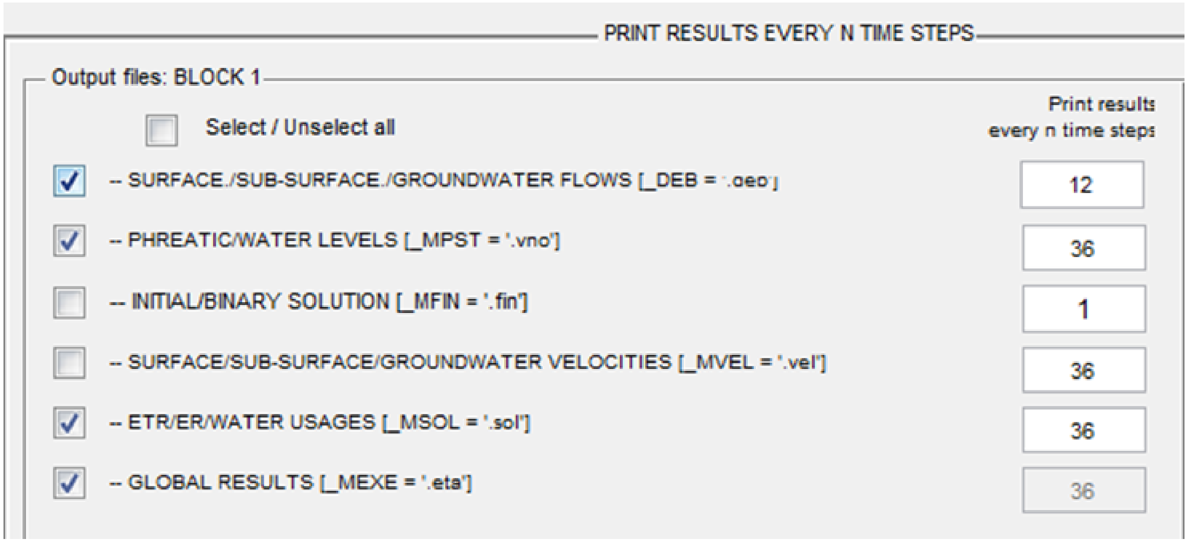


Figure 94: Configuration of the output files or model results.

See the following explanation about the output files and their configuration:

Water flows (DEB): check this box to print the subsurface, sub-soil and surface flows that pass through the gauging sections with a frequency of every (12) time steps. The flows printed are instantaneous and not average flows of the time lapse, therefore it is necessary to print the flows with a high frequency (low number of time steps) to obtain good hydrological water balances.

Water levels (VNO): check this box to print series of results related to free water levels, with a print frequency of every (36) time steps. The printed results in this file can be used to generate an initial solution of the simulation.

Initial solutions (FIN): uncheck this box if you prefer to built afterwards, if needed, an initial solution with the results of the water levels (VNO).

Water velocities (VEL): Check this box to print the subsurface, sub-soil and surface velocities fields at all nodes of the mesh. Define a print frequency of every (36) time steps.

Climatic variables and water usages (SOL): Check this box to print the average evaporation, transpiration and precipitation rates, as well as the flows rates of different uses of water in the time lapse given by the print frequency of every (36) time steps.

Global results (ETA): Check this box to print a series of global results necessary to verify the correct evolution of the simulation, the printing frequency is by default the same as that of the Water levels (VNO).

4.1.3.2 Preparing the Parameters/resolution

1. **Parameters of Surface Model (Figure 95)**:
   a. SURFICIAL X COEFFICIENT [-1 to 1]: define -0.935 to consider the influence of ground slopes (-) in the most common value of X.
   b. MINIMUM INTERACTION CONDUCTIVITY: define 1e-6 m/sec as the minimum conductivity for the influence of the ground slope on the X value, when it is considered (-).

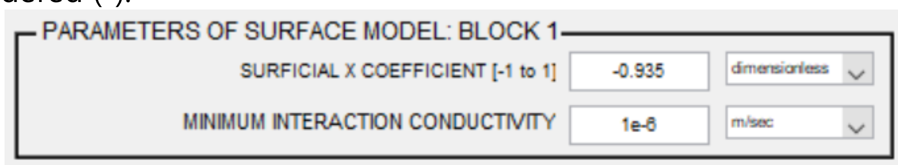


Figure 95: Surfface model parameter setting

2. **Parameters of sub-surface model (Figure 96):**
   a. SUB-SURFACE FACTOR: define -100 (to consider the influence of ground slope (-) in the most common value of the Sub-surface factor in order to smooth the hydrological properties in the subsoil zone.
   b. INFILTRATION GALLERY FACTOR: define 1 (to consider a fully infiltrating gallery, 0-1).
   c. MINIMUM SATURATED THICKNESS (Substratum): define 3 (as the minimum thickness, in meters, of a ground saturated medium, >0)
   d. SOIL/ROOTS THICKNESS: define 5 (as the global mean thickness of the soil, in meters). This value would be zonally defined in the .SOI file.
   e. CAPILLARY FRINGE THICKNESS: define 0.5 (as the global mean capillary height in the soil, in meters). This value would be defined by zones in the .SOI file.
   f. INITIAL RAW (field capacity): define 0 (as the initial water content in the subsoil zone, when the field capacity is defined by zones en the .SOI file, otherwise would be the global mean field capacity, 0-1).

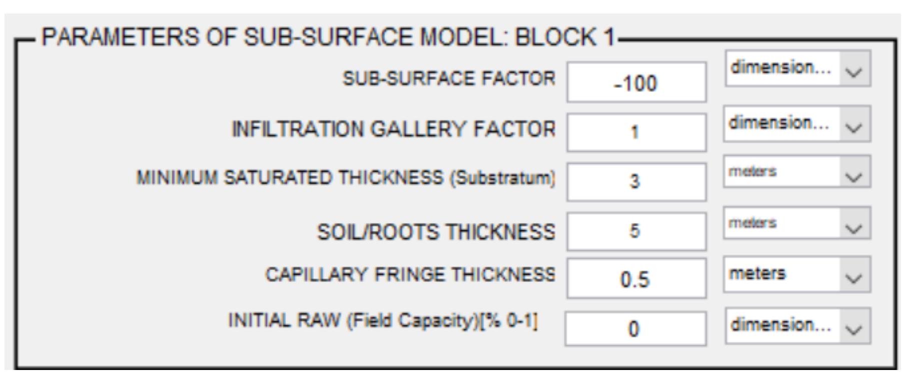


Figure 96: Sub-surface model parameter setting

3. **Parameters of floodgate / spillway (Figure 97):**

a. MAX/MIN CHANGE OF OVERFLOW LEVEL: define 4.5 (to consider, in meters, the máximun increase of the floodgates level, and besides the numerical decrease on the topographic level of pipe spillways in order to shun numerical instabilities.
b. CHANGE SPILLWAY FLOW: define -0.08 (to consider, in m3/sec, a constant reduction in the setting of spillway flow).
c. CHANGE FLOODGATE OPENING: define 0.01 (to consider, in meters, in this case a constant rise in the calculated levels that corresponds to the openings of the floodgate, +).

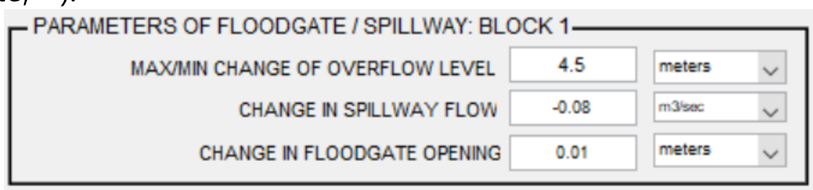


Figure 97: Floodgate / spillway parameter settings.

4. **Parameters of Loss of water global/zonal (Figure 98):**
   a. Precipitation / recharge: define 0 (to indicate that the precipitation/recharge should be define by zones in the SLR. file).
   b. Potential evapotranspiration: define 1 (>0, to indicate that the evapotranspiration model should be activated by zones with the parameters of the .SOI file, keep this value >0 as 1 mm/year).
   c. Potential evaporation: keep this value as 0 (to indicate that the evaporation model would be activated by zones with the parameters of the .SOI file).
   d. Global rate of Surface diversion: keep this value as 0 (to indicate that the diversion model would be activated by zones with the parameters of the .SLR file).
   e. Maximum diversion volume: define 10000 (>0, a very high value, in hm3, to indicate that you are not curbing the diversion model to be activated and working by zones with the parameters of the .SLR file).

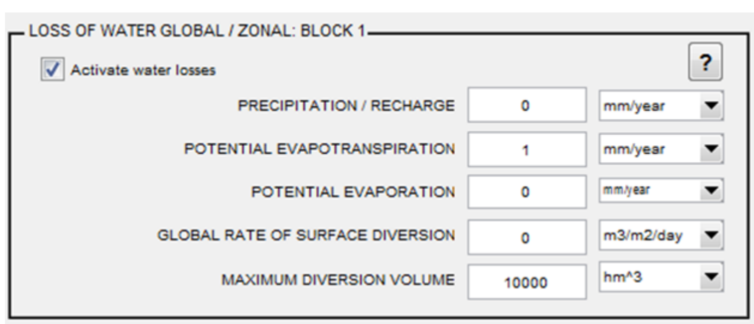


Figure 98: Global loss of water parameter setting.

5. **Parameters of Saltwater model (Figure 99):**
   a. Factor of Ghyben Herzberg: do not check on this dialog box, MELEF-FSW fresh/saltwater model for coastal watersheds is only available as a prototype and it is not still available for application on this Interface.

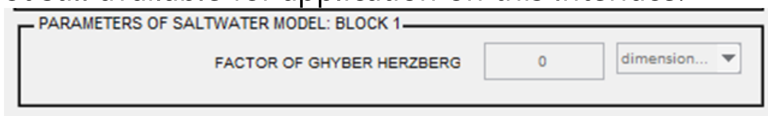


Figure 99: Saltwater model parameter setting.

6. **Parameters of System resolution (Figure 100):**

MELEF-FSW uses a preconditioned iterative algorithm GMRES, one of the best numerical solvers that allows a simple processing of the numerical mesh.

   a. MODEL OF FRESH/SAL WATER: This application is a continental watershed, so only freshwater would be considered, keep the option of Continental (AL2H). Continental-Coastal (AL2S) model is only available as a prototype, and it is not still available for application on this Interface.
   b. STEADY/TRANSIENT RESOLUTION: keep the EULR resolution which implies a transient simulation.
   c. CONVERGENCE ERROR (_EPSDL): keep this value around 1.0E-15, numerically has proven right.
   d. PRECONDITIONING NUMBER (NPREC): keep this value 2, numerically has proven enough (>1).
   e. ITERATION NUMBER (NRDEM): keep this value 20, numerically has proven adequate.
   f. SIZE NUMBER (NITER): keep this value 25, numerically has proven enough in most cases.
   g. PRECONDITIONER (_IML): keep this value 1 (Mass), numerically has proven better.

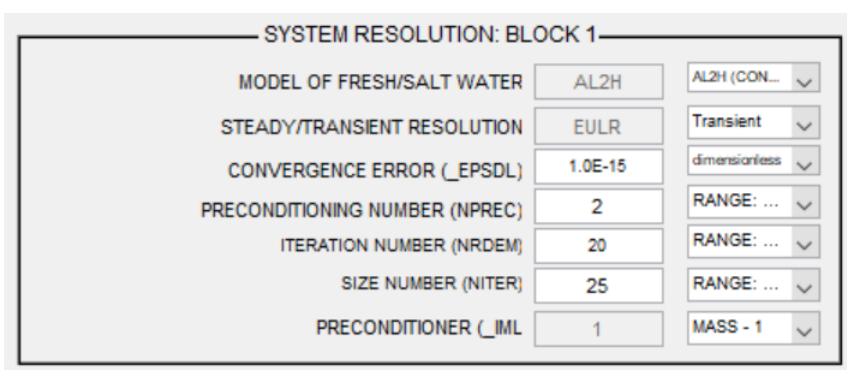


Figure 100: GMRES System resolution parameter settings.

4.1.3.3 Preparing the transient database .XLS

Select in the menu *Properties* of the User Interface the *Transient Database* tool, and generate the transient databases for the GIS files SLRMELEF.dbf and SOIMELEF.dbf. Finally, these two databases are filled with information that represent transient behavior of the simulation conditions and water uses.

The tool reads the vector attributes table (points or nodes of the mesh), first to validate it, and second, reads the different labels of the *TIME_FILE* column to generate the Transient DataBase in a XLS format.

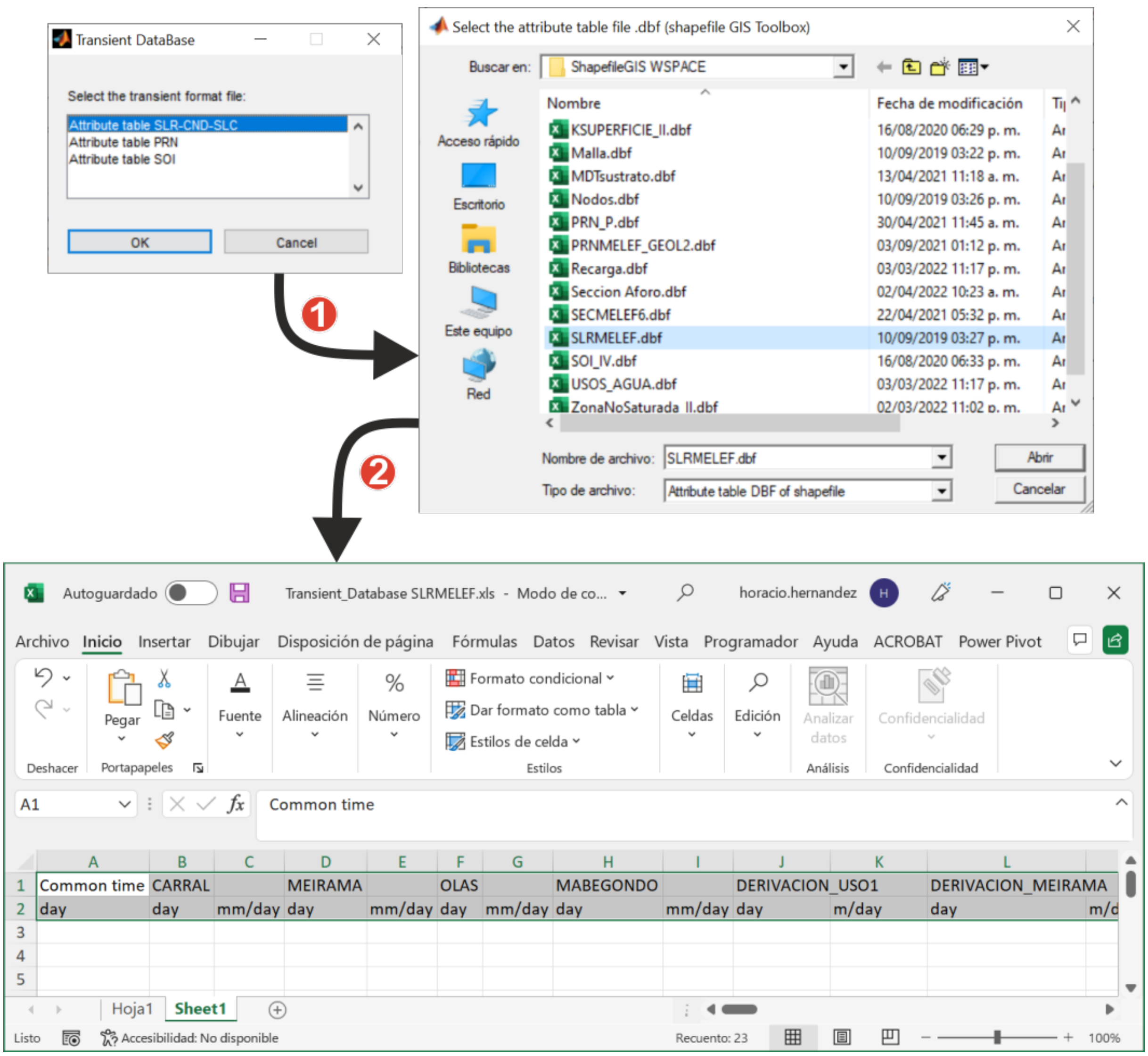


Figure 101: Process to create a transient database (XLS) using as input a GIS shapefile attribute table generated with the GIS Toolbox.

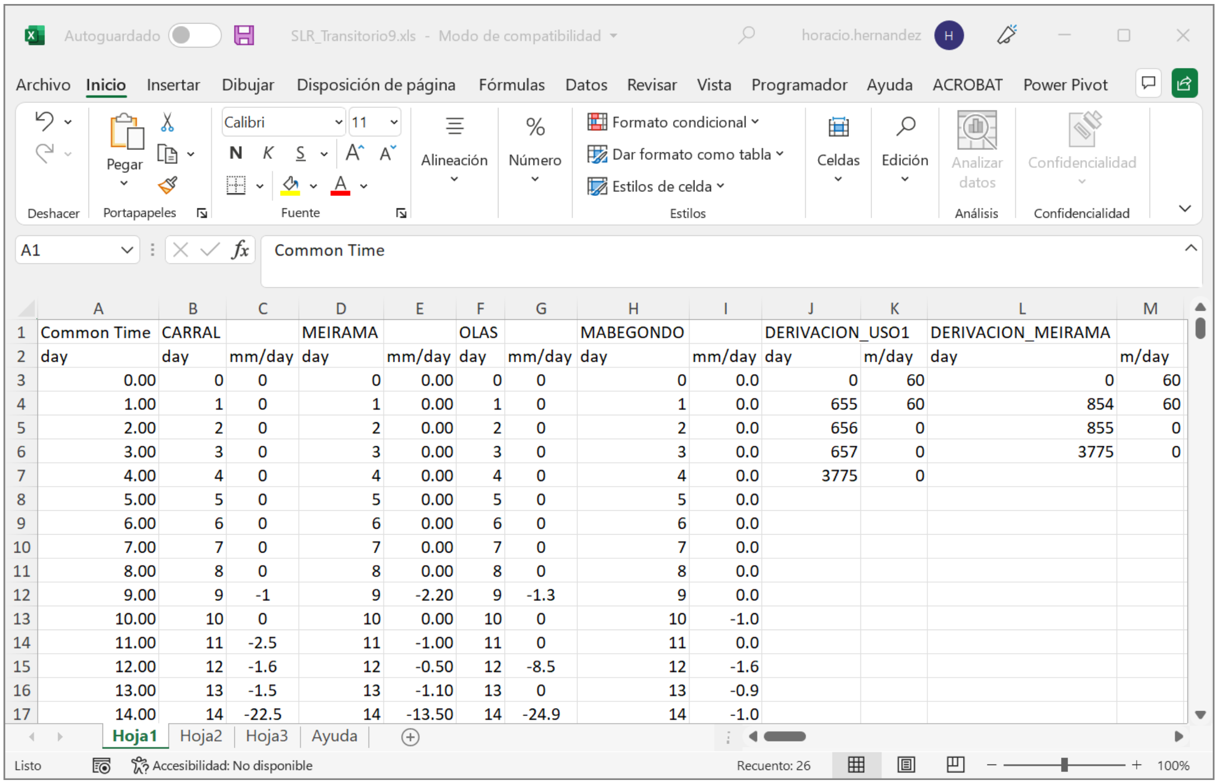
Follow the instructions shown in Figure 101:

1 Open the tool *Transient Database*. In the pop-up window list select the type of GIS simulation file you want to use as input file to generate their transient database in a XLS format, press *OK*.

2 Define the name file and output path to save the Transient Database. By default, the User Interface opens the Transient Database folder to save here the file.

The next step is to open the Excel file to fill with the observed or acquired information, follow the description of Figure 102 by the location of rows and columns in the file:

ROW 1: This row contains the TIME_FILE tags from the GIS simulation files.
ROW 2: This row contains the time and value measurement units with the magnitude of water uses and simulation conditions.
A1: Common Time, it is a column intended to define the increment of time required and total duration to be printed in the SLR simulation file. It is independent of the rest of columns and it is used to interpolate the magnitudes of water uses and simulation conditions to the time increment defined in this column.
B1: TIME_FILE label "CARRAL", it is a name of a rainfall station where from ROW 3 onwards this column is filled with time information in days (B2) from the start date of the simulation.
C1: TIME_FILE label "MEIRAMA", it is a name of a rainfall station where from ROW 3 onwards this column is filled with time information in days (C2) from the start date of the simulation.
L1: TIME_FILE label "DERIVACION_MEIRAMA", it is a name of the simulation condition *Surface Water Diversion* applied at the Meirama open pit, and the rate of the maximum water diversion is in the column M in units of m/day (unit that is equivalent to m3/m2/day).



	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Common Time	CARRAL		MEIRAMA		OLAS		MABEGONDO		DERIVACION_USO1		DERIVACION_MEIRAMA	
2	day	day	mm/day	day	mm/day	day	mm/day	day	mm/day	day	m/day	day	m/day
3	0.00	0	0	0	0.00	0	0	0	0.0	0	60	0	60
4	1.00	1	0	1	0.00	1	0	1	0.0	655	60	854	60
5	2.00	2	0	2	0.00	2	0	2	0.0	656	0	855	0
6	3.00	3	0	3	0.00	3	0	3	0.0	657	0	3775	0
7	4.00	4	0	4	0.00	4	0	4	0.0	3775	0		
8	5.00	5	0	5	0.00	5	0	5	0.0				
9	6.00	6	0	6	0.00	6	0	6	0.0				
10	7.00	7	0	7	0.00	7	0	7	0.0				
11	8.00	8	0	8	0.00	8	0	8	0.0				
12	9.00	9	-1	9	-2.20	9	-1.3	9	0.0				
13	10.00	10	0	10	0.00	10	0	10	-1.0				
14	11.00	11	-2.5	11	-1.00	11	0	11	0.0				
15	12.00	12	-1.6	12	-0.50	12	-8.5	12	-1.6				
16	13.00	13	-1.5	13	-1.10	13	0	13	-0.9				
17	14.00	14	-22.5	14	-13.50	14	-24.9	14	-1.0				

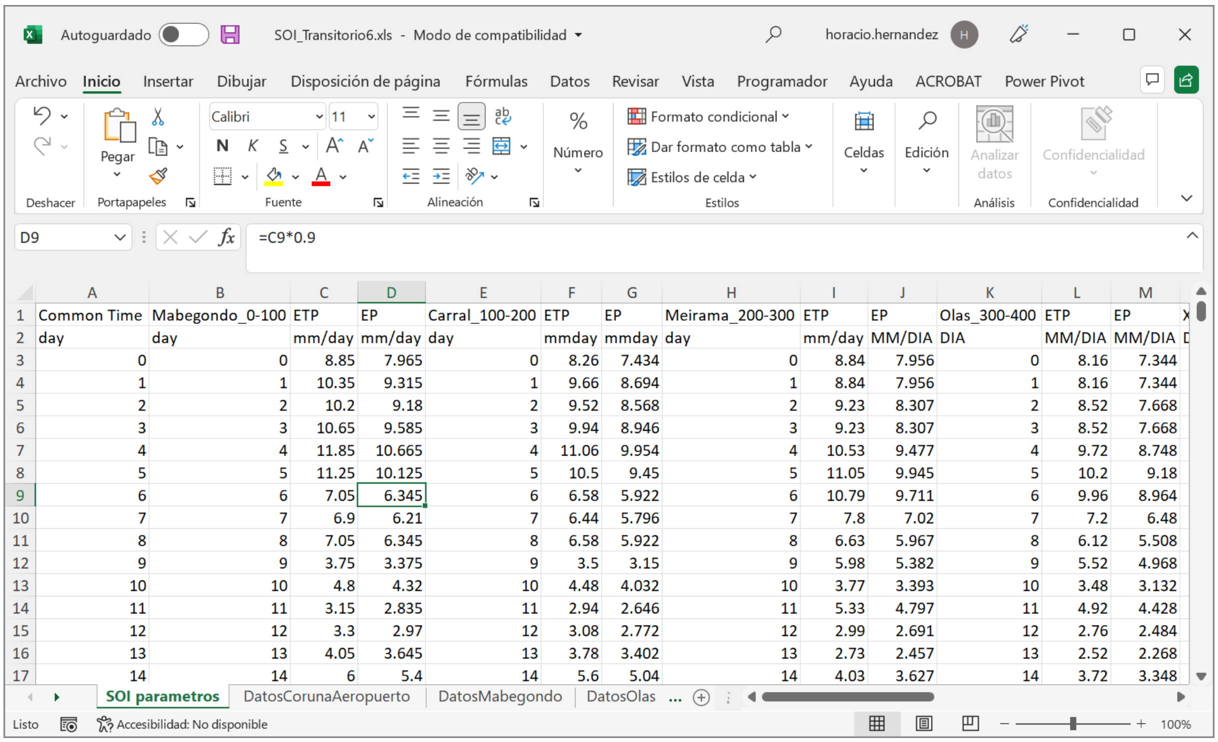
Figure 102: Transient database XLS created with the tool Transient Database and using as input file the GIS attribute table SLRMELEF.dbf. It is an example with filled values.

The next example is how to fill the Transient Database for the simulation file SOI, follow the description of Figure 103 by the location of rows and columns in the file:

ROW 1: This row contains the TIME_FILE tags from the GIS simulation files followed by the variables with a transient behavior.
ROW 2: This row contains the time and value measurement units with the magnitude of climate variables and soil properties with a transient behavior.
A1: Common Time, it is a column intended to define the increment of time required and total duration to be printed in the SLR simulation file. It is independent of the rest of columns and it is used to interpolate the magnitudes of water uses and simulation conditions to the time increment defined in this column.
B1: TIME_FILE label "Mabegondo_0-100", This column is the name of a zone where there are more than one parameter whose transient behavior can be defined, these parameters are shown after this column and are ETP and EP, columns C1 and D1 respectively.
E1: TIME_FILE label "Carral_100-200", This column is the name of a new zone where there are more than one parameter whose transient behavior can be defined, these

parameters are shown after this column and are ETP and EP, columns F1 and G1 respectively.

As can be seen, the pattern of printing zones defined by the labels in the TIME_FILE field is repeated, where each label is preceded by the variables it contains. Moreover, in this transient database, the times used always start from zero, where zero means the initial time of the simulation. The User Interface adds the simulation start date to these times if you need to view the results in date format.



	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Common Time	Mabegondo_0-100	ETP	EP	Carral_100-200	ETP	EP	Meirama_200-300	ETP	EP	Olas_300-400	ETP	EP
2	day	day	mm/day	mm/day	day	mmday	mmday	day	mm/day	MM/DIA	DIA	MM/DIA	MM/DIA
3	0	0	8.85	7.965	0	8.26	7.434	0	8.84	7.956	0	8.16	7.344
4	1	1	10.35	9.315	1	9.66	8.694	1	8.84	7.956	1	8.16	7.344
5	2	2	10.2	9.18	2	9.52	8.568	2	9.23	8.307	2	8.52	7.668
6	3	3	10.65	9.585	3	9.94	8.946	3	9.23	8.307	3	8.52	7.668
7	4	4	11.85	10.665	4	11.06	9.954	4	10.53	9.477	4	9.72	8.748
8	5	5	11.25	10.125	5	10.5	9.45	5	11.05	9.945	5	10.2	9.18
9	6	6	7.05	6.345	6	6.58	5.922	6	10.79	9.711	6	9.96	8.964
10	7	7	6.9	6.21	7	6.44	5.796	7	7.8	7.02	7	7.2	6.48
11	8	8	7.05	6.345	8	6.58	5.922	8	6.63	5.967	8	6.12	5.508
12	9	9	3.75	3.375	9	3.5	3.15	9	5.98	5.382	9	5.52	4.968
13	10	10	4.8	4.32	10	4.48	4.032	10	3.77	3.393	10	3.48	3.132
14	11	11	3.15	2.835	11	2.94	2.646	11	5.33	4.797	11	4.92	4.428
15	12	12	3.3	2.97	12	3.08	2.772	12	2.99	2.691	12	2.76	2.484
16	13	13	4.05	3.645	13	3.78	3.402	13	2.73	2.457	13	2.52	2.268
17	14	14	6	5.4	14	5.6	5.04	14	4.03	3.627	14	3.72	3.348

Figure 103: Transient database XLS created with the tool Transient Database and using as input file the GIS attribute table SOIMELEF.dbf. It is an example with filled values.

4.1.4 Practice 2: Preparing GIS simulation layers

Objectives of the practice:

- Import the geometry of the triangular finite element mesh into ArcMap.
- Generate the GIS files of simulation and edit the simulation conditions with the help of the MELEF Toolbox. Generate the shapefiles that will store the simulation conditions and with the editing tool draw polygons or poly-lines around or on the nodes that have a specific simulation condition.
- Create the GIS files needed to Generate the input files of the MELEF-FSW model for the user Interface.

NOTES: To start this practice open the GIS software QGIS or ArcMap. For the case of ArcMap, it is recommended to define the *Workspace* and *Scratch Workspace* folders in *Geoprocessing>Environment...* menu and select a directory to save the output and temporal layers.

4.1.4.1 Importing the mesh to GIS

1. Import the geometry of the mesh into GIS (QGIS, ArcMap):
   a. Open the tool "*1.1 Import Nodes - Mesh (*.GEO - *.2DM . - *.MSH)*"
      i. Select the file with the mesh in a format .GEO - .2DM or .MSH. For this case, select the file MallaCMero_Sept_25_2012.2dm from the path *...Mero2020/Simulaciones MELEF*
      ii. It is recommended to select a Digital Elevation Model (DEM) in raster format to evaluate the slope in each node of the mesh.
      iii. Define a Coordinate Reference System to avoid possible reference problems with the output layers.
      iv. It is recommended to maintain the original names of the output layers [Mesh; Nodes; DTMsubstratum, Catchment] to avoid confusion in other tools that require these layers.
      v. This tool generates the following layers that are necessary to define the simulation conditions:
         1. Nodes.shp: Layer of points with the mesh nodes, the nodes ID, the slope and influence area.
         2. Mesh.shp: Layer of polygons with the triangular elements of the mesh.
         3. Basin.shp: Layer of polygons with the study area.
         4. DTMsubstratum.shp: Layer to store elevations of topography and impervious substratum.
2. The tool itself defines the layers it requires, see Figure 104.

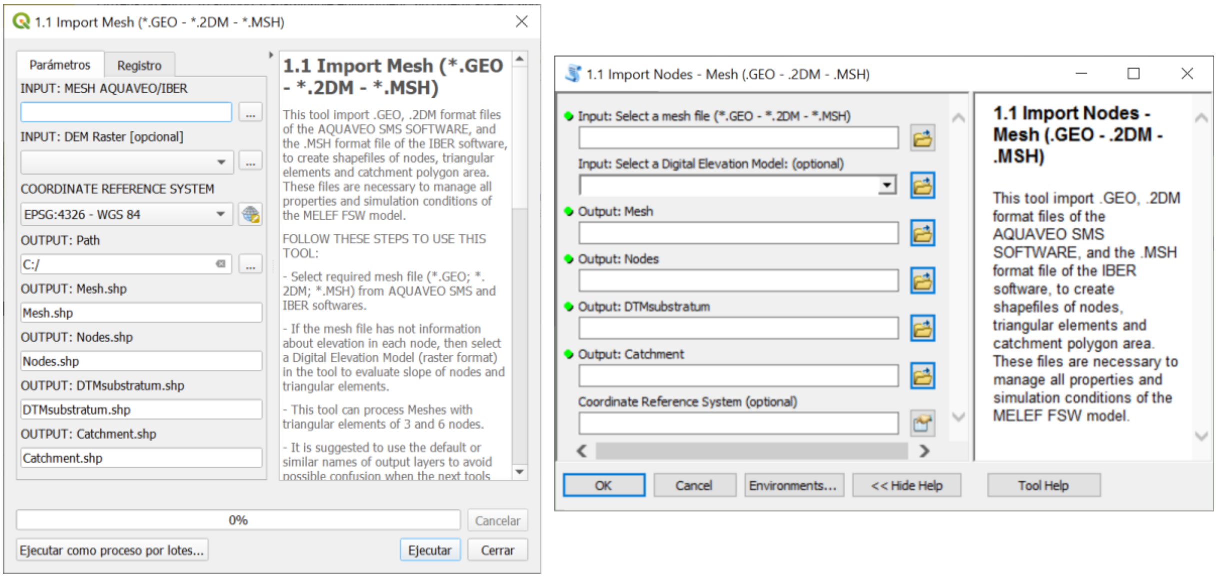


Figure 104: Tool to import the mesh and create GIS layers (QGIS, ArcMap).

4.1.4.2 Create SOIMELEF with soi properties and EvapoTranspiration rates

1. Previously to execute the tool "2.3 Generate SOIMELEF" you need to:
   a. Create a "2.1 New Non Saturated Zone"
   b. Create a "2.2 New Ksuperficial"
2. The equivalent layers of the step 1 in the folder MERO_FONDO>Mero 2020>ShapefileGIS WSPACE are:
   a. ZonaNoSaturada_GradAdiab2.shp = NonSaturatedZone.shp
   b. KSUPERFICIE.shp = Ksuperficial.shp
   c. nodos.shp = Nodes.shp
3. If you need to modify the layers, edit the shapefile. If the layers are empty, you can copy polygons from an existing layer with information about soil and vegetation types.
4. Select the layers of the step 2 in the tool to create the layer SOIMELEF.shp, see Figure 105.

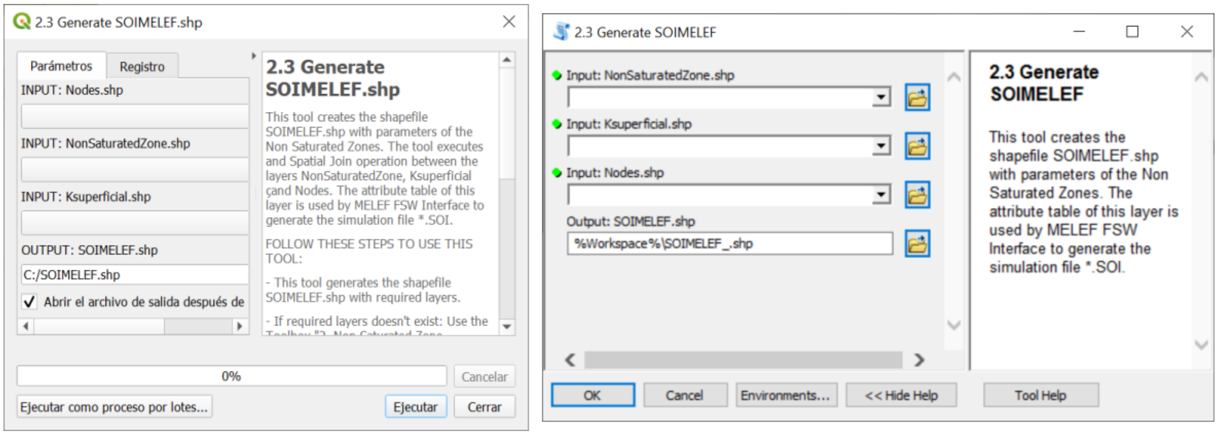


Figure 105: Tool to generate the SOIMELEF layer with the information required to construct the simulation file SOI in the user interface.

4.1.4.3 Create PRNEMELF with geological material properties

1. Previously to execute the tool "[3.3 Generate PRNMELEF]" you need to:
   a. Define in the DTMsubstratum the location of the impervious substratum. "[3.1 Approximate Impervious Substratum]".
   b. Create a "[3.2 New Geology]"
2. The equivalent layers of the step 1 in the folder MERO_FONDO>Mero 2020>ShapefileGIS WSPACE are:
   a. GEOLOGIA.shp = Geology.shp
   b. MDTsustrato.shp = DTMsubstratum.shp
3. If you need to modify the layers, edit the shapefile. If the layers are empty, you can copy polygons from an existing layer with geological materials. Manage KX, KY, KZ material properties and ALPHA anisotropy angle from the shapefile attribute table.
4. Select the layers of the step 2 in the tool to create the layer PRNMELEF.shp, see Figure 106:

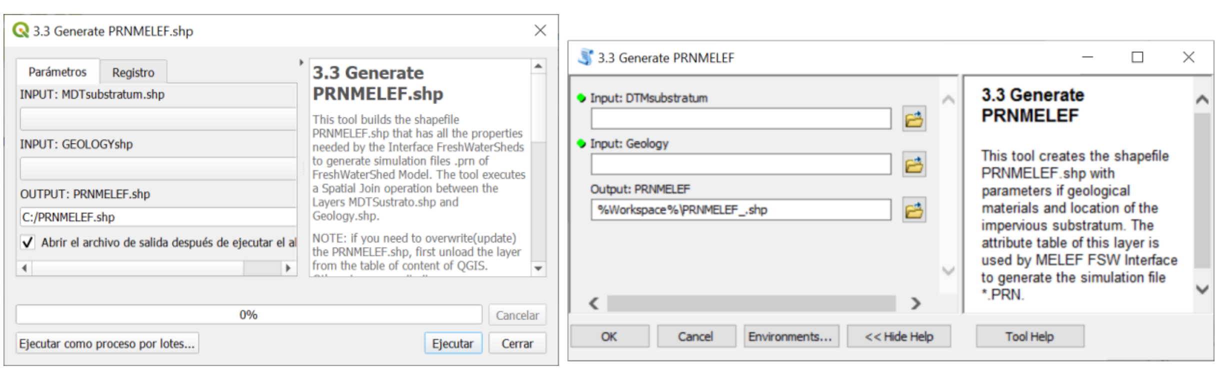


Figure 106: Tool to generate PRNMELEF layer with the gelogical material properties to construct the PRN simulation file.

4.1.4.4 Create CNDMELEF/SLCMELEF with boundary conditions.

1. Previously to execute the tool "4.2 Generate CNDMELEF or SLCMELEF" you need to:
   a. Create a "4.1 New CND - SLC"
2. The equivalent layers of the step 1 in the folder MERO_FONDO>Mero 2020>ShapefileGIS WSPACE are:
   a. CND.shp = CND.shp
   b. nodos.shp = Nodes.shp
3. If you need to modify the layers, edit the shapefile. If the layers are empty, you can copy polygons from an existing layer with boundary or lateral conditions.
4. Select the layers of the step 2 in the tool to create the layer CNDMELEF.shp or SLCMELEF.shp, see Figure 107.

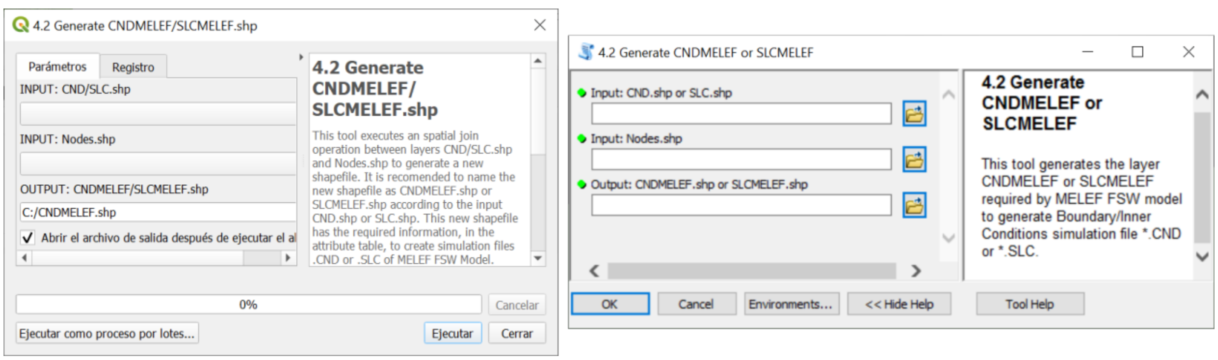


Figure 107: Tool to generate the GIS layer file CNDMELEF or SLCMELEF required by the user interface to generate the simulation file CND or SLC.

4.1.4.5 Create SLRMELEF with the rainfall and water uses.

1. Previously to execute the tool "5.2 Generate SLRMELEF" you need to:
   a. Create a "5.1 New Rainfall - Water Uses"
2. The equivalent layers of the step 1 in the folder MERO_FONDO>Mero 2020>ShapefileGIS WSPACE are:
   a. Recarga_Altura.shp =Rainfall.shp
   b. uso_agua.shp = WaterUses.shp
3. If you need to modify the layers, edit the shapefile. If the layers are empty, you can copy polygons from an existing layer with rainfall zones or water uses. In the attribute table of the shapefile (Rainfall) confirm that the field COD_USO (code of use) is the value 5, as well as the shapefile Water Uses has in the attribute table field COD_USO values of 4, 6, 8 and 9.
4. Select the layers of the step 2 in the tool to create the layer SLRMELEF.shp, see Figure 108:

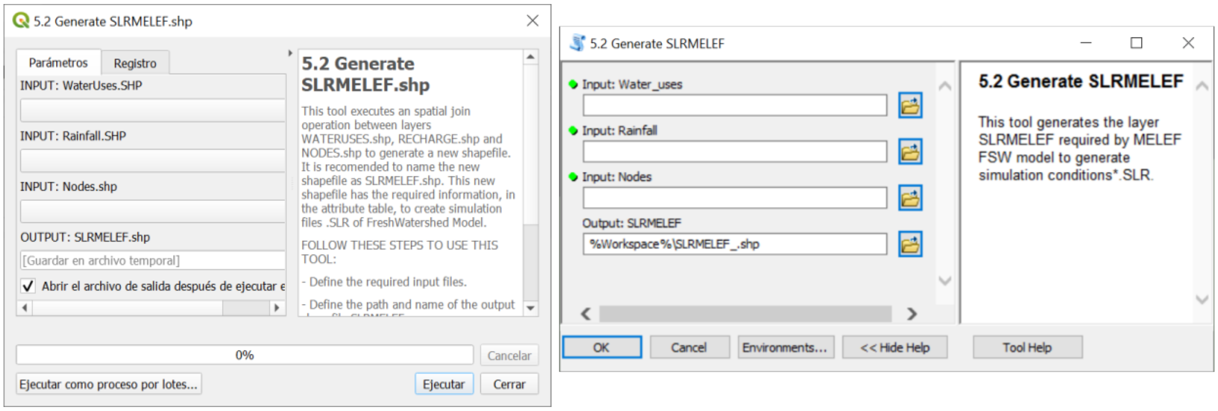


Figure 108: Tool to generate the SLRMELEF GIS layer required by the user interface to generate the simulation file SLR.

4.1.4.6 Create SECMELEF with the gauging sections

1. Previously to execute the tool "6.2 Generate SECMELEF" you need to:
   a. Create a "6.1 New GaugeSection"
2. The equivalent layers of the step 1 in the folder MERO_FONDO>Mero 2020>ShapefileGIS WSPACE are:
   a. SeccionesAforo.shp = GaugeSection.shp
3. If you need to modify the layers, edit the shapefile to copy polylines features from an existing layer or manually generate the polylines based on some available information where the gauging section would be setted.
4. Select the layers of the step 2 in the tool to create the layer SECMELEF.shp, see Figure 109.

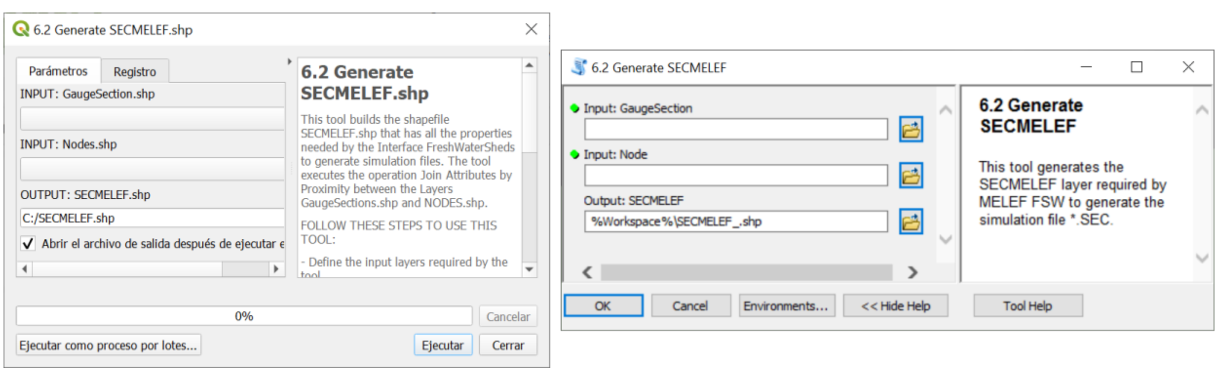


Figure 109: Tool to generate the GIS layer SECMELEF with the required information to generate the simulation file SEC in the user interface.

4.1.5 Practice 3: Preparing and running MELEF FSW

The aim of this practice is to explain the steps required to prepare and run the MELEF FSW. As well as, the importance of prepare the model every time there is a modification in a simulation file or with the configuration of simulation conditions and system resolution.

Go to the panel *Simulation control* and then to the section *2. RUN CONTROL OF MELEF FSW MODEL,* on this section select the in the *Simulation folders* the folder *Mero 6hrs Nuevo 2*, the follow the next steps in Figure 110 that are explained in the following lines:

1: Click on the button *PREPARE* to copy the simulation files to the execution folder before running the model. This step updates any changes to the simulation files or changes to the INP file with the simulation conditions, global parameters and system resolution that the model needs to execute.

2: Message in dialog box that appears if there are previous simulation files with the same name. Click the *Continue* button to define which files should be updated.

3: By default all simulation files that are enabled for simulation are selected for preparation, you can choose which files should be prepared and click on the *Continue* button.

4 CASE 1: the execution folder has not files of results with the same name and then an MS-DOS window appear with information about the model, with the % of execution and the convergence error (NORME). The NORME is a good indicator of the numerical resolution process, where NORME <1 implies a good convergence value. You can check during the model run the results of the simulation process in the *Results* menu of the MELEF-FSW Interface. Check also with the  button the live information about the estimated time for the run process to finish. At any time, you can stop/cancel the run process by pressing Ctrl+C in the MS-DOS window. The execution can be restarted as long as the appropriate changes are made to the simulation files (reading leap).

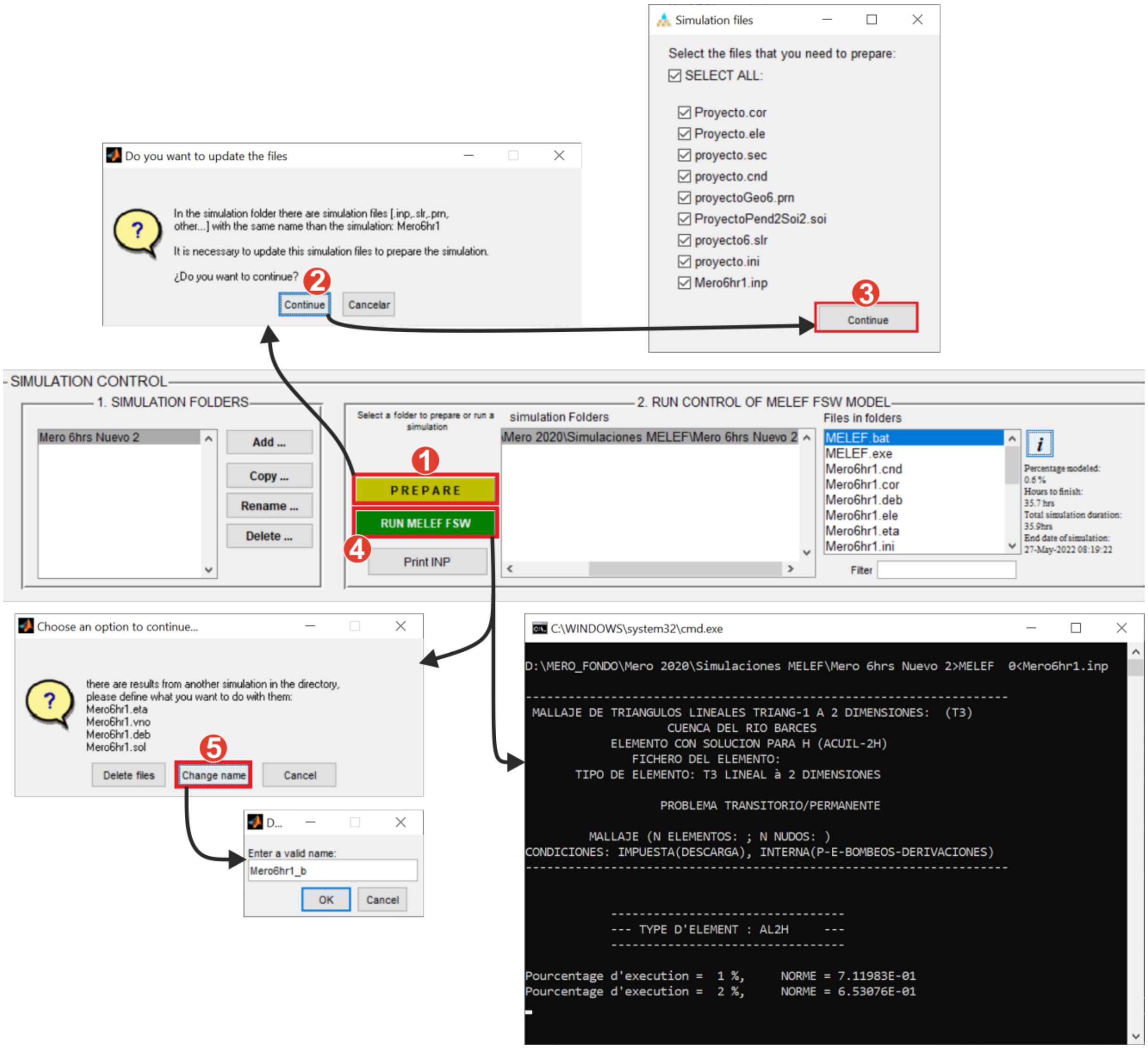


Figure 110: Process to prepare and execute the MELEF FSW model.

4.2 Activate Fresh/Salt Water Model

Consider the following steps in the order given to activate the saltwater intrusion simulation or continental/coastal interaction model.

USER INTERFACE:

1. Go to the [Panel 3: Parameters / resolution]() and change in the *System Resolution Block* the model of fresh/saltwater from AL2H(Continental) to A2HS (Continental - Coastal).
2. After activate the model A2HS go to the variable *Factor of Ghyben Herzberg*, located in the same *Panel 3*, and define a value. The range of values for this variable is [25 - 35] and it depends on the specific weights of salt water and

freshwater. See the technical manual in the section of *Groundwater Model* for more details about the formulation of *Ghyben Herzberg*.

GIS TOOLBOX:

3. Boundary Condition: create a CNDMELEF shapefile with a boundary condition located where the source of saltwater is, and use the *Code of Use* defined for fresh/saltwater model interaction [11, 21, 41]. The attribute table of the CNDMELEF has several columns including [VAL_PERM; VALPERM2], these two columns are intended to save the permanent value of the imposed boundary condition for fresh and saltwater respectively.
   a. Code of Use 11: it is recommendable to start with this code of use and use as permanent values [VAL_PERM; VAL_PERM2] the elevation zero for fresh and saltwater level.
   b. Code of Use 21: use this condition to impose an in(-)/out(+) diffuse lateral salt water flow (m3/sec) in the boundary.
   c. Code of Use 41: use this code of use to evaluate the slope of the beach where the factor value can be [0, 1], where [0] automatically evaluates the slope of the beach but it can create numerical instability, and [1] is the recommendable factor to start because considers an horizontal slope of the beach.

USER INTERFACE:

4. Go to the *Panel 2: Simulation conditions* and create a new simulation file with the tool *Create SLR/CND/SLC* using as input the CNDMELEF.dbf generated in the step 3. If you need to impose the sea water tide (code of use 11) as a transient boundary condition, then use the tool *Transient Database* to generate an .XLS file, using as input the CNDMELEF.dbf generated in the step 3, with the required columns structure and fill with the time (in days starting where the initial date of simulation is the day one) and the tide elevation (fill the VAL_PERM and VAL_PERM2 with the same information).
5. Create an initial solution (*Simulation files* section in *Panel 3*) clicking the button *Define...* and in the window that appears check the option -*Print column with saltwater initial solution*- and consider the following options:
   a. Constant elevation: select this option if there is not a previous initial solution.
   b. From file first: select this option to update the initial solution with a a .fin file result generated in a previous simulation. NOTE: print the output file *Initial/Binary Solution [_MFIN = *.fin]* to create future initial solution.
6. Go to the *Panel 1: Simulation control* and prepare the simulation to run the MELEF FSW model with fresh/saltwater interaction.

Index of Figures

Figure 1: Hydrologic representation of groundwater and surface, continental and coastal water flows, solved by the MELEF FSW model. 12
Figure 2: Results analysis capabilities of MELEF FSW User Interface. 13
Figure 3: Toolbox MELEF FSW versions for QGIS and ArcMap of ESRI. 14
Figure 4: Diagram with the software required for management of a new Project. 15
Figure 5: Menu "Windows>Catalog" to open the catalog window in ArcMap. 18
Figure 6: Path to the Toolbox MELEF for ArcMap of ESRI. 18
Figure 7: Steps to add the GIS Toolbox to QGIS. 19
Figure 8: GIS Toolbox for QGIS after import the folder "Scripts QGIS 3x" 20
Figure 9: Most relevant steps for a simulation project with MELEF FSW. 24
Figure 10: Project Manager window. 26
Figure 11: Project Menu: New Project 26
Figure 12: The path and name of the open project is displayed in the main window. 26
Figure 13: Sub directories generated to manage a simulation project. 27
Figure 14: Description of different sections of the Panel 1: Simulation control. 28
Figure 15: Panel 2: Use this panel to define the start and end simulation time, generate the simulation files and configurate result files. 30
Figure 16: Dialog box to define the simulation files. 31
Figure 17: Window for defining the initial solution for the water table, the readily available water in the soil, and the location of salt water. It is possible to define a variable solution from a VNO-FIN-SOL results file, a constant solution, or, in the case of the water table, there is a third option, which consist of using the topography to generate a water table parallel to it. 33
Figure 18: Panel 3: Use this panel to configurate global parameters, activate/deactivate models and modify the system resolution. 36
Figure 19: Description of the configuration and results analysis tools contained in the menu toolbar of the MELEF Interface. 41
Figure 20: Tool to import the mesh and generate the simulation files .COR and .ELE. 42
Figure 21: Tool to import the DBF of a the shapefile SECMELEF created with Toolbox GIS and create simulation file with the gauging sections. 43

Index of Figures

Figure 22: Tool to generate the geological and topographic nodal properties of the discrete model. The tool implements the attribute table DBF file of the shapefile PRNMELEF generated with the GIS Toolbox. 44
Figure 23: Tool to generate the soil nodal properties of the discrete model. The tool implements the attribute table DBF file of the shapefile SOIMELEF generated with the GIS Toolbox. 47
Figure 24: Tool to generate the simulations files SLR, CND or SLC. The tool implements the attribute table DBF file of the shapefile SLRMEEF, CNDMELEF, SLCMELEF generated with the GIS Toolbox. 50
Figure 25: Tool to create the transient database structure in an Excel file using the attribute table DBF file associated with the shapefile generated with the GIS toolbox. 52
Figure 26: Tool to analyze the global results in the ETA output file of the model. 54
Figure 27: Tool to analyze the flows in the gauging sections of the model. 59
Figure 28: Settings of lines and markers of the tool Flows DEB. Furthermore, in this settings you can modify the percentage of Qsubalve that is added to Qgroundwater or Qsurfacewater. 61
Figure 29: Tool to export flow rates of the DEB file. 63
Figure 30: Nodal results tool for analysis of VNO-SOL-VEL simulation files. 64
Figure 31: Map to identify the location, magnitud and number of the nodes, it also includes contour lines to identify areas by topography. 65
Figure 32: Tool for zonal analysis of SOL output files printed by MELEF FSW by zones defined in a shapefile layer (this tool uses the DBF file associated to the shapefile layer). 70
Figure 33: Map displayed to identify nodal properties and find out zones. 72
Figure 34: Results visualization tool, this tool creates hydrological vertical profiles and maps of velocity vectors. 73
Figure 35: Tool to generate videos with results 2D: VNO - SOL 77
Figure 36: Tool to execute balances in areas completely delimited by gauging sections. This tool requires the selection of multiple files for analysis. 80
Figure 37: Select the required files in the multiple file selection window. 81
Figure 38: Graph with the water balance performed by the tool Zonal Balance. 84

Index of Figures

Figure 39: Graph with the materials change in storage evaluated by Balance and expected by groundwater levels in materials. 85
Figure 40: Tools to import the mesh in QGIS (Left) and ArcMap (Right). 86
Figure 41: Tools to create a new empty layer NonSaturatedZone with the predefined attribtue table structure in QGIS and ArcMap. 88
Figure 42: Tools to create a new empty layer Ksuperficial with the predefined attribute table structure in QGIS and ArcMap. 89
Figure 43: Tools to generate the simulation layer SOIMELEF in QGIS and ArcMap. 90
Figure 44: Tools to aproximate an impervious substratum in QGIS and ArcMap. 92
Figure 45: Tools to generate the new empty layer Geology with a predefined attribute table structure in QGIS and ArcMap 93
Figure 46: Tools to generate the simulation layer PRNMELEF in QGIS and ArcMap. 94
Figure 47: Tools to create a new empty layer CND or SLC with the attribute table structure predefined in QGIS and ArcMap. 95
Figure 48: Tools to generate the simulation layers CNDMELEF and SLCMELEF in QGIS and ArcMap 97
Figure 49: Tools to generate the new empty layer Rainfall/WaterUses with the attribute table structure predefined in QGIS and ArcMap. 97
Figure 50: Tools to generate the simulation layer SLRMELEF in QGIS and ArcMap. 99
Figure 51: Tools to create the new empty layer GaugeSection with the attribute table structure predefined in QGIS and ArcMap. 100
Figure 52: Tools to generate the simulatin layer SECMELEF in QGIS and ArcMap. 101
Figure 53: Tool to adjust the maximum infiltration capacity of the soil (Kz) in ArcMap. 102
Figure 54: Evolution of the adjusted Soil Thickness using different parameter configuration of the vegetation factor Fveg. 103
Figure 55: Tool to adjust the Soil Thickness in function of Slope and a vegetation factor. 103
Figure 56: Tool to create the simulation file COR in ArcMap. 105
Figure 57: Tool to generate the simulation file .INI with the initial solution for the model MELEF FSW. 106
Figure 58: Representation of the Dem Burning Process by Slope over a stream profile. 107
Figure 59: Tool to run the DEM Burning by Slope process and recondition a DEM raster to create hydrologically correct DEM. 108

Index of Figures

Figure 60: Tool to execute the process of DEM Burning Elevation for recondition of a DEM raster and create a hydrologically correct raster. 109
Figure 61: Representation of the Agree Method variables and influence over a stream section. 110
Figure 62: Tool to execute the DEM Recondition Multiple process and re-define the riverbed sections perpendicular to the streams path. 110
Figure 63: Tool to fill sinks/depressions in the Digital Elevation Model. 112
Figure 64: Tool to generate buffers along streams and control the transition between areas with high to low density of mesh nodes. 113
Figure 65: Tool to redistribute the distance between vertex of polyline or polygon layer. 114
Figure 66: Tool to generate a polygon layer with all the depressions evaluated by the tool. 115
Figure 67: Tool to delete the Extra Columns generated by some geoprocessing tools in the output layer that are not in the source layers. 116
Figure 68: Tool that update a field of a point layer with the information from a raster layer. 117
Figure 69: Salt/Surface/Groundwater model interaction of MELEF FSW model. 143
Figure 70: Surface/groundwater flow interaction in MELEF FSW. Thickness of groundwater (Zgw), subsoil (Zss) and stream flow (Zs). Actual EvapoTranspiration (ETa). 146
Figure 71: Working scheme of a sluice floodgate 151
Figure 72: Sketch for small dam with pipe spillways and opening valves. 154
Figure 73: Actual evapotranspiration uptake method interaction. 157
Figure 74: Soil water balance to define disconinuous ETa and the groundwater Net Recharge. 160
Figure 75: Interaction between underground and surface models, as well as porosity and transmissivity changes. 161
Figure 76: Sketch for tunnels and galleries condition and their activation in GIS 162
Figure 77: Automatic injection zone managed through the field "ZONAINYEC" in the WaterUses layer of GIS. 166
Figure 78: Automatic irrigation zone managed through the field "ZONAINYEC" in the WaterUses layer of GIS. 167
Figure 79: Convergence implications of reduce/increase the ratio of model parameter fractions. 169

Index of Figures

Figure 80: Representation of the interpolation process as a histogram with backward step. 174
Figure 81: Multiple file selection window. 175
Figure 82: Tool to configurate the video recording settings. 176
Figure 83: Digital Elevation Model that needs a reconditioning of the terrain elevations to make it hydrologically correct. 179
Figure 84: Flooding of the Meirama open pit (LIMEISA, 2009). 185
Figure 85: Mero catchment location (~ 247 km2). North-Western Spain. S1-S9 and RG1-RG5 are the locations of measured discharge sections and Rain Gauging of MeteoGalicia Climate Stations, respectively. 186
Figure 86: Geology of the Mero catchment area: zoom in the Meirama open pit mine zone 187
Figure 87: Finite elements mesh model and rainfall zones in the Mero River catchment area. Water management and boundary conditions at the Meirama open pit (A) and the Cecebre Reservoir (B, C). 189
Figure 88: Water management in the outlet of the dam of the Cecebre Reservoir. Openings of pipe spillways and floodgates. 190
Figure 89: Open the project of Mero River Basin with the practical case of the flooding of the Meirama open pit. 192
Figure 90: Process to define the simulation files if the status box is black. 193
Figure 91: Process to define an existen initial solution file. 194
Figure 92: Configuration of the Time Control for the project Malla21-09-2012_nuevoPerfilPresa 195
Figure 93: Define the name of the simulation files once these are copied to the execution folder. 195
Figure 94: Configuration of the output files or model results. 196
Figure 95: Surfface model parameter setting 197
Figure 96: Sub-surface model parameter setting 197
Figure 97: Floodgate / spillway parameter settings. 198
Figure 98: Global loss of water parameter setting. 198
Figure 99: Saltwater model parameter setting. 199
Figure 100: GMRES System resolution parameter settings. 199
Figure 101: Process to create a transient database (XLS) using as input a GIS shapefile attribute table generated with the GIS Toolbox. 200
Figure 102: Transient database XLS created with the tool Transient Database and using as input file the GIS attribute table SLRMELEF.dbf. It is an example with filled values. 202

Index of Figures

Figure 103: Transient database XLS created with the tool Transient Database and using as input file the GIS attribute table SOIMELEF.dbf. It is an example with filled values. 203
Figure 104: Tool to import the mesh and create GIS layers (QGIS, ArcMap). 205
Figure 105: Tool to generate the SOIMELEF layer with the information required to construct the simulation file SOI in the user interface. 206
Figure 106: Tool to generate PRNMELEF layer with the gelogical material properties to construct the PRN simulation file. 206
Figure 107: Tool to generate the GIS layer file CNDMELEF or SLCMELEF required by the user interface to generate the simulation file CND or SLC. 207
Figure 108: Tool to generate the SLRMELF GIS layer required by the user interface to generate the simulation file SLR. 208
Figure 109: Tool to generate the GIS layer SECMELEF with the required information to generate the simulation file SEC in the user interface. 208
Figure 110: Process to prepare and execute the MELEF FSW model. 210

Index of Tables

Table 1: Table with suggested minimum CPU characteristics 17
Table 2: Table with basic units of the MELEF FSW 172
Table 3: Table with units that can be used in Spanish 173
Table 4: Table with units that can be used in English 173
Table 5: Table with standard date formats to use in Excel/CSV databases. 174
Table 6: Table the code of use and condiions required to activate the simulation conditions, water uses and boundary conditions 182

Index

- A -

actual EvapoTranspiration 159
Actual simulation 30
Add block 35
Adjust ES by Slope and Vegetation 102
Adjust KZ by Slope 102
AGREE DEM 109
ALFA 44, 93
Animations 1D VNO-VEL 73
Animations 2D VNO-SOL-VEL 76
APPLICATION AND TRAINING 184
Approximate an Impervious Substratum 90
Aquaveo SMS 14
ArcGIS 12
Automatic water bypass 163

- B -

Block settings 30
Boundary conditions 30
Boundary In/Outflow Volume 53
Boundary Out/Inflow 53

- C -

calibration 168
Capillary Fringe Thickness 46, 87
Cathy 12
Cattaneo model 143
CCAMPO_PM 46, 87
Change in spillway flow 35
Change to floodgate opening 35
Coastal Basin Area 53
Coastal FreshGroundwater Volume 53
Coastal SaltGroundwater Volume 53
Coastal SurfaceWater Volume 53
Code of use 181
CODIGO_USO 49, 95
Continental Basin Area 53
Continental Groundwater Reserve 53
Continental SurfaceWater Reserve 53
Continuous evaporation and transpiration 156
Convergence Error 35
COTA1 44
COTA2 44
Create CNDMELEF/SLCMELEF 207
Create or open a simulation project 25
Create PRN 44
Create PRNEMELF 206
Create SEC 43
Create SECMELEF 208
Create SLR/CND/SLC 49
Create SLRMELEF 207
Create SOI 46
Create SOIMELEF 205
Curve: Area – Perim.Wet 59
Curve: Depth – AreaSection 59
Curve: Depth – Perim.Wet 59
Curve: Qsurface – AreaSection 59
Curve: Qsurface – Depth 59
Curve: Qsurface – Perim.Wet 59
Curved vector 73

- D -

define gauge sections 100
Delete Extra Fields 115
DEM Burning by Slope 106
DEM Burning Elevation 108
DEM Reconditioning Multiple 109
Depth Surface Water 76
Depth to SaltWater Interface 76
Depth to Water Table 76
Differential model 143
Diffusion wave hydrodynamic model 143
Discontinuous transpiration 158
Diverted Water Volume 53
Draw the gauging section 100

- E -

EC 46, 87
Effective porosity 44
End date of simulation 30
EP 46, 87
EPSDL 35
ES 46, 87
ETP 46, 87
Evapo(Transpiration) thickness 53

Execute FreshSaltWatershed 27
Execution and results files 27
Execution folders paths 27
Export to ASCII for GIS 76

- F -

Factor of infiltration gallery 35
Files name 30
Fill Sinks in DEM 111
Filter files 27
Floodgates simulation 150
Flows DEB 59
Formulations ground/surface models 140
Fresh/Saltwater Model 35
Freshwater 141

- G -

Gauge sections ground/surface 30
Generate animation 73
Generate Buffers to Create Mesh Zones 112
Generate CNDMELEF or SLCMELEF 96
Generate PRNMELEF 94
Generate SECMELEF 101
Generate simulation file COR 104
Generate simulation file INI 105
Generate SLRMELEF 99
Generate SOIMELEF 90
Ghyben-Herzberg factor 35
GIS CODE OF USE 181
GIS Toolbox in ArcMAP 16
GIS Toolbox in QGIS 16
Global ETA 53
Global Evapo(Transpiration) 53
Global Evapo(Transpiration) Volume 53
Global Precipitation 53
Global Precipitation thickness 53
Global Precipitation Volume 53
Global Rate of Surface Diversion 35
Groundwater Model 141
groundwater net recharge 159
GroundWater Pumping 53

- I -

IBER 2D Hydraulic 14
IML 35
Import Mesh 85
Importing the mesh to GIS 204
Initial solution 30
Input CDN file 120
Input COR file 120
Input ELE file 121
Input INI file 121
Input INP file 122
Input PRN file 124
Input SEC file 125
Input simulation files 120
Input SLC file 126
Input SLR file 127
Input SOI file 125
Installation and configuration 16
Installation Process 16
Interaction models 156
Interpolation as histogram with backward step 174
irrigation 163
Iteration number 35
IWFM Mesh Generator 14

- K -

KSUPERF 89
KSUPERFICIAL 46
KX 44, 93
KY 44, 93
KZ 44, 93

- L -

Lateral inflow on boundaries 53
Lateral Inflow Volume 53
Lateral stream conditions 30

- M -

Material properties 30
Matlab Runtinme 16
Max. Diversion Volume 35
Max/Min Change of overflow level 35
MELEF CODE OF USE 181
Menu Mesh 42
Menu Project 41
Menu Properties 43

Menus and Tools 41
Mesh Geometry 30
Mike Basin 12
Mike She 12
Minimum interaction conductivity 35
Minimum saturated thickness (substratum) 35
ModeFlow 12
Mosaic of plots 53
Multiple file selection window 175
Muskingum model 143

- N -

N 44, 93
New CND - SLC 95
New Gauge Section 100
New Geology 93
New Ksuperficial 89
New Non Saturated Zone 87
New Project 41
New Rainfall - Water Uses 97
NITER 35
NNUDO 44
Nodes VNO-SOL-VEL 63
Non Saturated Zone - EvapoTranspiration (SOI) 87
NPREC 35
NRDEM 35
Numerical conditions and resolution 160

- O -

Open Project 41
Output DEB file 130
Output ETA file 132
Output FIN file 137
Output SOL file 138
Output VEL file 139
Output VNO file 139
Overland flow model 159
OverlandFlow Area 53
OverlandFlow thickness 53

- P -

Panel 1: Simulation control 27
Panel 3: Parameters / resolution 210
Parameters / resolution 35
Percentage Ground SaltWater 76
Percentage Surface SaltWater 76
Potential Evaporation 35, 46
Potential EvapoTranspiration 35, 46
Practical case 1: Flooding of Meirama open pit mine (River Mero Basin) 184
Precipitation/Recharge 35
Preconditioner 35
Preconditioning number 35
Prepare simulation 27
Preparing and running MELEF FSW 208
Preparing GIS simulation layers 204
Preparing the Parameters/resolution 197
Preparing the Simulation conditions 191
Print INP 27
Printing the results 30
Process IWFM Mesh 43
Process Mesh (geo-2dm-msh) 42
Project Manager 41
Pumped Water Volume 53

- Q -

QGIS 12
Qgroundwater 59
QoverlandFlow 59
Qsubsoil 59
Qsurface 59

- R -

Readily Available Water 46, 87
Reading settings 30
Recharge 53, 132, 138
Reconditioning of a raste relevation model 177
Redistribute Vertex 113
Remaining run time 27
Required software 14
Reservoir operations 150
Result simulation files 130
results analysis capabilities 12
Rules of convergence 168

- S -

Saltwater 141
Saves Changes 41

Scientific publications 20
Section view 73
Simulation conditions 30
Simulation files 30
Simulation folders 27
Size number 35
SLOPE 46
Soil properties 30
Soil Thickness 46, 87
Soil water balance 159
Soil/Root thickness 35
Spillways simulation 152
Starting records/simulation 30
Steady/Transient Resolution 35
Straight vector 73
Sub-surface factor 35
Surface inflow on boundaries 53
Surface Inflow Volume 53
Surface Model 143
Surface model redefined 143
SurfaceWater Area 53
SurfaceWater Diversion 53
Surficial X coefficient 35
SUSTRATO 44

- T -

Thickness of capillary fringe 35
Thickness Salt Water 76
Thickness Water Table 76
Time step 30
TIME_FILE 44, 200
Toolbar description 27
Topographic Depressions Evaluation 114
Transient Database 52
Triangular Finite Element Mesh 14
Tunnels and Galleries simulation 161

- U -

UNIDADES 44
Units of measure 172
Update Field by Raster 116
usable information 24

- V -

VAL_PERM 49
Variation Qgrounwater 59
Variation Qsubsoil 59
Variation Qsurface 59
Vel.Groundwater 59
Vel.OverlandFlow 59
Vel.Subsoil 59
Vel.Surface 59
Velocity vectors 73
Video Recording 176

- W -

Water Injections 53
Water Injections Volume 53
Water inputs/loss/uses 30
Water Table Change 76

- X -

X Coefficient 36

- Z -

ZONA 44
ZONA_INYECCION 49
Zonal Balance 80
Zonal Precipitation 53
Zonal Precipitation thickness 53
Zonal Precipitation Volume 53
Zonal SOL 69
Zonal Transpiration 53
Zonal Transpiration thickness 53
Zonal Transpiration Volume 53

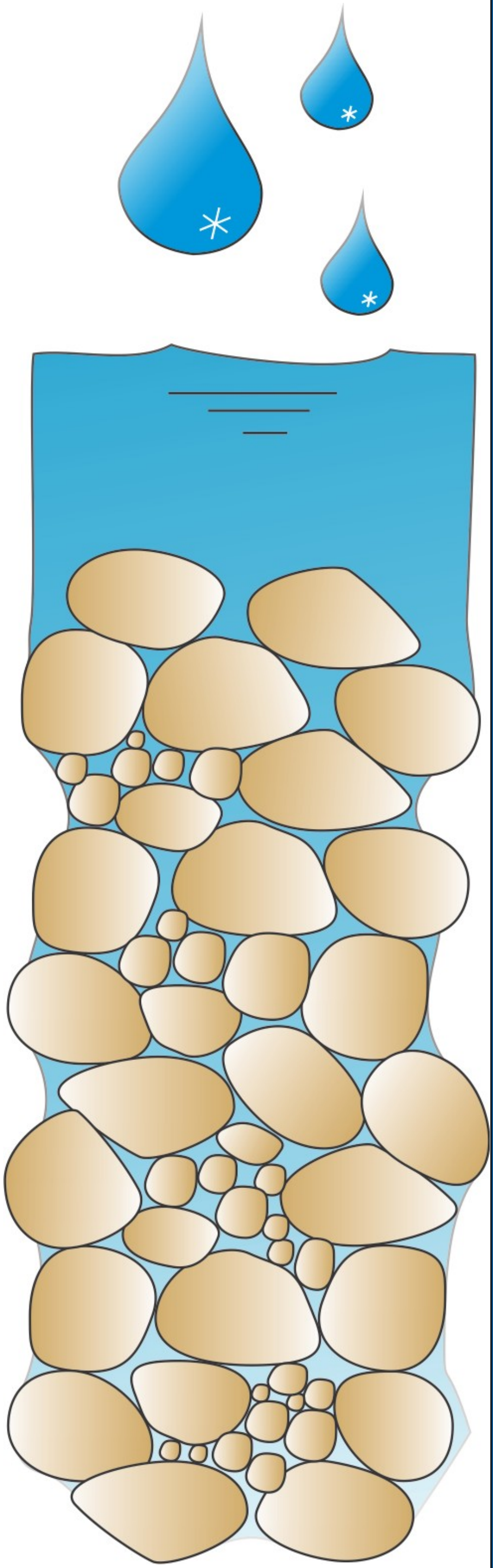
FUTURE VERSIONS OF THIS MANUAL

Future enhancements to the Graphical User Interface and the GIS Toolbox, both part of the FreshSaltWatershed, have as main objective to integrate further with the numerical code MELEF FSW.

This integration will make it possible to increasingly separate the use of the different programs to manage a simulation project.

This help manual, in its second version, documents the most used capabilities of FreshSaltWatershed code, as well as the Geographic Management System developed around it.

In future versions, other features will be included that have not been possible to include due to the variety of possibilities that MELEF FSW currently presents.



Help Manual MELEF FreshSaltWatershed

This manual compiles the second version of the help manual for MELEF-FreshSaltWatershed products: User Interface; GIS Toolbox; MELEF-FSW numerical model.

The reader will find in this help manual:

1. An introduction to the scope and capabilities of the MELEF FSW model.

2. A user manual with the main functions of the Graphical User Interface, a description of each of the pre- and post-process tools, as well as the internal structure of the database that manages all the Interface configuration.

2. A description of all the tools developed with python scripts to create the GIS Toolbox for QGIS and ArcMap, to manage the information generated by all simulation conditions and model boundary conditions.

3. A detailed description of the structure and content of all input files, output files or model results, as well as all simulation files coming from the User Interface and the GIS Toolbox.

4. A detailed introduction to the operation of the main simulation capabilities (simulation conditions) of the MELEF-FSW numerical model.

5. An example of an application, in tutorial form, on applied cases like Cecebre reservoir and the controlled flooding of the Meirama open pit mine as a lake (River Mero Basin, Galicia, Spain).

Dr. Horacio Hdez, is a full professor at the University of Guanajuato in Mexico, his main contribution to this project has been the design and development of the Graphical User Interface and GIS Toolbox, as well as important improvements to the MELEF-FSW numerical model and its application to solve environmental problems with groundwater and surface water interaction. His doctorate was carried out under the direction of Dr. Francisco Padilla, with the main objective of applying, improving and developing a MELEF FSW code management system.