

Processing Modflow **11**

Version 11.0.5

March 26, 2025



Copyright © 2025 Simcore Software

This Page Intentionally Left Blank



Contents

Acknowledgements.....	vi
Application and Data Folders.....	viii
Glossary	ix
Supported MODFLOW Packages.....	x
Variables	xi
Image of the Start Window.....	xii
Chapter 1. Introduction.....	1
1.1. What is Processing Modflow.....	1
1.2. Main Window of PM	2
1.3. Model Settings	7
Chapter 2. Maps.....	11
2.1. Views	11
2.2. Captions.....	12
2.3. Image Overlays.....	13
2.4. Raster Maps.....	14
2.5. Base Maps	16
2.5.1. Edit Base Maps.....	16
2.5.2. Customize the Display of a Base Map.....	18
2.6. Shapefiles	19
2.6.1. Add Shapefiles to the Map Viewer	19
2.6.2. Customize the Display of a Shapefile.....	19
Chapter 3. Modeling Environment.....	25
3.1. Units	25
3.2. Temporal Discretization	25
3.3. Spatial Discretization.....	26
3.4. Grid Editor and Data Editor	27
3.4.1. Grid Editor	27
3.4.2. Data Editor	29
3.5. File Menu.....	50
3.5.1. New	50
3.5.2. Open.....	51

3.5.3.	Recent Files	51
3.5.4.	Save	51
3.5.5.	Save As	51
3.5.6.	Save As Reduced Size	51
3.5.7.	Create Model	52
3.5.8.	Import Model	54
3.5.9.	Save Image	56
3.5.10.	Animation.....	57
3.5.11.	Print.....	58
3.5.12.	Preferences	58
3.6.	Tools Menu.....	64
3.6.1.	Pan Tool	64
3.6.2.	Zoom Tool	65
3.6.3.	Observation Tool.....	65
3.6.4.	MODPATH Tool	73
3.6.5.	Measure Tool	73
3.6.6.	Row Cross Section Tool.....	74
3.6.7.	Column Cross Section Tool	76
3.6.8.	Pin Tool	76
3.6.9.	Rotation Tool.....	77
3.6.10.	Location Tool.....	77
3.6.11.	Finish Edit.....	77
3.6.12.	Cell-by-Cell Tool	77
3.6.13.	Polyline Tool.....	77
3.6.14.	Polygon Tool.....	78
3.6.15.	Pilot Points Tool	78
3.6.16.	Cell Copy.....	78
3.6.17.	Layer Copy.....	78
3.6.18.	Grid Size Copy	78
3.6.19.	Grid Split Copy.....	78
3.6.20.	Tools > Data Menu	78
3.6.21.	Tools > Package Menu	86
3.7.	Discretization Menu	88
3.7.1.	Time	88

3.7.2.	Grid.....	90
3.7.3.	Layer Property.....	90
3.7.4.	Layer Top Elevation and Layer Bottom Elevation.....	93
3.7.5.	IBOUND (Flow).....	94
3.7.6.	ICBUND (Transport).....	94
3.8.	Parameters Menu.....	95
3.8.1.	Starting Hydraulic Head.....	95
3.8.2.	Horizontal Hydraulic Conductivity (HK) and Transmissivity (T).....	95
3.8.3.	Horizontal Anisotropy (HANI).....	95
3.8.4.	Vertical Hydraulic Conductivity (VK) and Vertical Leakance (VCONT).....	96
3.8.5.	Vertical Anisotropy (VANI).....	96
3.8.6.	Specific Storage (Ss) and Storage Coefficient (S).....	96
3.8.7.	Specific Yield (Sy).....	97
3.8.8.	Effective Porosity (ne).....	97
3.8.9.	Stop Zone.....	97
3.8.10.	Retardation Factor (R).....	97
3.8.11.	Bulk Density.....	98
3.8.12.	Static Sandboxes.....	98
3.8.13.	Transient Sandboxes.....	98
3.9.	Models Menu.....	98
3.9.1.	MODFLOW.....	98
3.9.2.	ZoneBudget.....	208
3.9.3.	MT3D/SEAWAT.....	209
3.9.4.	GSFLOW.....	237
3.9.5.	PEST.....	266
Chapter 4.	Visualization and Postprocessing.....	309
4.1.	Input Data.....	309
4.1.1.	Model Grid.....	309
4.1.2.	Packages.....	311
4.1.3.	Observation Points.....	311
4.2.	Visualization.....	312
4.2.1.	Input Parameter Contours.....	312
4.2.2.	Result Contours.....	318

4.2.3.	Flow Vectors.....	322
4.2.4.	Particle Tracking with MODPATH	326
Chapter 5.	Supplementary Information	335
5.1.	File Formats.....	335
5.1.1.	2D Matrix and 2D Weight Matrix.....	335
5.1.2.	2D Modflow Array.....	335
5.1.3.	3D Modflow Array.....	336
5.1.4.	Comma-Separated Values File	337
5.1.5.	ASCII Raster	337
5.1.6.	Shapefile.....	339
5.1.7.	Particle File.....	339
5.1.8.	Georeference file	340
5.1.9.	Observation File	341
5.1.10.	PM Pilot Points File	344
5.1.11.	Polygon File	345
5.1.12.	Polyline File	349
5.1.13.	Reservoir Stage Time Series.....	355
5.1.14.	Series File	355
5.1.15.	SURFER ASCII GRD File	356
5.1.16.	Tabular Inflow File.....	357
5.1.17.	Temporal Discretization File	358
5.1.18.	Well File.....	359
5.2.	Utilities	362
5.2.1.	MOD2OBS	362
5.2.2.	KRIGING.....	362
5.2.3.	FHBADJUST.....	363
5.2.4.	UPDATEWEIGHTS.....	363
5.2.5.	LAYERWEIGHT	364
5.2.6.	LAKEGAGE	365
5.2.7.	STREAMGAGE.....	365
5.2.8.	REAL2MATRIX	366
5.3.	Input and Output Data Files of the Supported Models and Utilities	366
5.3.1.	MODFLOW	366

5.3.2.	MT3DMS/MT3D-USGS	368
5.3.3.	GSFLOW	368
5.3.4.	SEAWAT.....	369
5.3.5.	MODPATH	369
5.3.6.	PEST.....	370
5.3.7.	PPK2FAC1	371
5.3.8.	FAC2REAL	372
5.3.9.	PPCOV	372
5.3.10.	PPKREG1.....	372
References	373
Index	381

Acknowledgements

The development and evolution of Processing Modflow would not have been possible without the invaluable support, dedication, and inspiration from countless individuals, institutions, and vibrant open-source communities.

The journey of Processing Modflow began in 1991 with the release of its first version. Over the decades, it has evolved into a comprehensive and versatile groundwater modeling software. I express my sincere gratitude to the original developers of MODFLOW, whose groundbreaking work has significantly advanced groundwater modeling and hydrogeological research globally. Special recognition goes to the dedicated teams at the U.S. Geological Survey, whose continuous innovations and unwavering commitment to maintaining MODFLOW as an accessible, open-source tool have profoundly benefited the scientific community.

I also extend heartfelt appreciation to the authors and developers of essential complementary modeling tools. Their outstanding contributions have vastly enhanced the capabilities of Processing Modflow. Specifically, I thank the developers of PEST for pioneering advancements in parameter estimation and uncertainty analysis; MT3DMS and MT3D-USGS for their indispensable work in solute transport modeling; MODPATH for particle-tracking simulations; GSFLOW for integrating groundwater and surface water modeling; SEAWAT for advancing variable-density flow and transport modeling; BeoPEST and PESTPP for enabling high-performance computational approaches to parameter estimation; and Zonebudget for their crucial developments in water budget analysis. Collectively, these powerful tools have empowered Processing Modflow users to tackle increasingly complex groundwater challenges with confidence.

Processing Modflow's success also stands firmly upon the robust foundations provided by the open-source software ecosystem. I sincerely thank the developers and contributors of Qt, VTK (The Visualization Toolkit), SQLite, GDAL (Geospatial Data Abstraction Library), and PROJ4, whose libraries have greatly enriched our software's capabilities in visualization, database management, geospatial processing, and coordinate transformations.

I owe profound gratitude to Professor Emeritus Wolfgang Kinzelbach, whose mentorship and deep expertise in hydrology and groundwater modeling have significantly shaped my professional journey. His guidance and encouragement have been pivotal to the success of this project. I warmly thank Geologisches Landesamt Hamburg, where I began my professional career, gaining essential experiences and skills that laid the groundwork for my path in groundwater modeling. Additionally, my heartfelt appreciation goes to Mark Wildermuth, my former employer, colleague, and friend, whose unwavering support, trust, and encouragement have greatly influenced my professional growth.

Special thanks go to my colleagues, collaborators, and beta testers, whose invaluable feedback, dedication, and meticulous testing significantly improved both the functionality and user experience of Processing Modflow. Their insights and commitment have been instrumental in shaping the software to meet the evolving needs of the hydrological community.

To the countless users of Processing Modflow—consulting firms, universities, government agencies, and individual professionals—your continued financial support and constructive feedback have been crucial to sustaining the software’s development and enhancements. Your active engagement and trust have enabled Processing Modflow to remain a dynamic tool, serving the diverse and ever-changing needs of the global groundwater modeling community.

Lastly, my deepest gratitude goes to my family and friends for their unwavering patience, understanding, and encouragement throughout this long and rewarding journey. Their support has been a cornerstone of my perseverance and commitment.

Thank you to everyone who has contributed to the development, advancement, and success of Processing Modflow.

Eric Wen Hsing Chiang

Irvine, California

Application and Data Folders

The following folders will be created and used by Processing Modflow. Replace YourUserName with your Windows username.

Application Files

Per-user installation (with pmx64_11.0.xx.exe):

C:\Users\YourUserName\AppData\Local\Simcore Software\Processing Modflow 11

Per-machine installation (with pmx64admin_11.0.xx.exe) for all users:

C:\Program Files\Simcore Software\Processing Modflow 11

Simulation Model Executable Files

Per-user installation (with pmx64_11.0.xx.exe):

C:\Users\YourUserName\AppData\Local\Simcore Software\Processing Modflow 11\Engine

Per-machine installation (with pmx64admin_11.0.xx.exe) for all users:

C:\Program Files\Simcore Software\Processing Modflow 11\Engine

Application Cache and Temporary Files:

C:\Users\YourUserName\AppData\Simcore Software\Processing Modflow

Examples: See readme.txt in the zip files for details

<https://simcore.com/files/pm/v11/examples/tutorials.zip>

<https://simcore.com/files/pm/v11/examples/basics.zip>

<https://simcore.com/files/pm/v11/examples/geo.zip>

<https://simcore.com/files/pm/v11/examples/lake.zip>

<https://simcore.com/files/pm/v11/examples/swi2.zip>

<https://simcore.com/files/pm/v11/examples/seawat.zip>

Glossary

ASCII Raster: ASCII Raster, also known as ASC/INFO ASCII GRID is a non-proprietary raster GIS file format developed by ESRI. See Section 5.1.5 for details.

CSV File: A comma-separated values file stores tabular data (numbers and text) in plain text. Each line of the file is a data record. Each record consists of one or more fields, separated by commas.

Ctrl+click, Shift+click, or Alt+click: Place the mouse pointer to an object; hold down the Ctrl (Shift or Alt) key; then press and release the left mouse button.

Ctrl+drag, Shift+drag, or Alt+drag: Hold down the Ctrl (Shift or Alt) key and the left mouse button, then move the mouse pointer.

Drag and drop or drag: Move the mouse pointer to an object; press and hold down the left mouse button; drag the object to the desired location by moving the mouse pointer; then drop the object by releasing the left mouse button.

EPSG: EPSG Geodetic Parameter Dataset is a public registry of geodetic datums, spatial reference systems, Earth ellipsoids, coordinate transformations and related units of measurement, originated by a member of the European Petroleum Survey Group (EPSG) in 1985. Each entity is assigned an EPSG code between 1024 and 32767 along with a standard machine-readable well-known text (WKT) representation.

GIS: A Geographic Information System (GIS) is a computer system that analyzes and displays geographically referenced information.

PM: Processing Modflow.

Shapefile: The shapefile format is a geospatial vector format describing spatial features such as points, polylines, and polygons. See Section 2.6 for details.

TOC: The Table of Contents window that is a part of the main window of Processing Modflow. TOC contains a list of predefined groups; each group may contain user-defined objects.

UTM: Universal Transverse Mercator.

WGS 84: World Geodetic System 1984. The corresponding EPSG code is 4326.

Mercator: The Mercator (also known as World Mercator) projection is a conformal cylindrical map projection. The corresponding EPSG code is 3395, which is equivalent to the code 54004 defined by Environmental Systems Research Institute (ESRI).

Supported MODFLOW Packages

Abbreviation	Package Name
BAS6	Basic
BCF6	Block-Centered Flow
CHD	Time Variant Specified Head
DE4	Direct Solver
DIS	Discretization
DRN	Drain
DRT	Drain with Return Flow
EVT	Evapotranspiration
ETS1	Evapotranspiration with a Segmented Function
FHB	Flow and Head Boundary
GAGE	Gage
GHB	General-Head Boundary
GMG	Generic Multigrid Solver
HFB6	Horizontal-Flow Barrier
IBS	Interbed-Storage
LAK3	Lake
LPF	Layer-Property Flow
LMT7	Flow-Transport Link
MNW1	Multi-Node Well 1
MNW2	Multi-Node Well 2
MNWI	Multi-Node Well Information
NWT	Newton Solver
OC	Output Control
PCG	Preconditioned Conjugate Gradient Solver
PCGN	PCG Solver with Improved Nonlinear Control
RCH	Recharge
RES1	Reservoir
RIV	River
SFR2	Streamflow-Routing
SIP	Strongly Implicit Procedure Solver
STR	Stream
SUB	Subsidence and Aquifer-System Compaction
SWI2	Seawater Intrusion 2

Abbreviation Package Name

UPW	Upstream Weighting
UZF1	Unsaturated-Zone Flow
WEL	Well

Variables

HANI	Horizontal anisotropy, dimensionless. $HANI \times HK$ = horizontal hydraulic conductivity along model columns.
HK	Horizontal hydraulic conductivity along model rows, in the unit of $[LT^{-1}]$.
m	Thickness of a model layer, in the unit of $[L]$.
ne	Effective porosity, dimensionless.
NCOL	Number of columns of a rectilinear (structured) MODFLOW model grid, dimensionless.
NLAY	Number of layers of a rectilinear (structured) MODFLOW model grid, dimensionless.
NROW	Number of rows of a rectilinear (structured) MODFLOW model grid, dimensionless.
NPER	Number of stress periods, dimensionless.
R	Retardation factor [-].
S	Storage coefficient, dimensionless.
Ss	Specific storage, in the unit of $[L^{-1}]$.
Sy	Specific yield, dimensionless.
T	Transmissivity, in the unit of $[L^2T^{-1}]$.
VANI	Vertical anisotropy, dimensionless. $VANI = HK / VK$.
VCONT	Vertical leakance, in the unit of $[T^{-1}]$.
VK	Vertical hydraulic conductivity, in the unit of $[LT^{-1}]$.
VKCB	Vertical hydraulic conductivity of a quasi-3D confining bed, in the unit of $[LT^{-1}]$.

Image of the Start Window



The start window of Processing Modflow features a scenic image of Hot Creek, situated in picturesque Mono County, California. Originating high in the eastern Sierra Nevada mountains as Mammoth Creek, this beautiful waterway is primarily fed by melting snow at an elevation of approximately 8,500 feet (2,600 meters) above sea level. As Mammoth Creek descends and transitions into Hot Creek, it becomes renowned not only for its breathtaking landscape but also for its geothermal activity, creating a unique and striking natural environment cherished by visitors and researchers alike.

Picture taken on June 21, 2019

The supported model codes include MODFLOW-2005 (Harbaugh, 2005), MODFLOW-NWT (Niswonger, et al., 2011), MODFLOW-OWHM (Boyce, et al., 2020), MODPATH (Pollock, 2016), MT3D-USGS (Bedekar, et al., 2016), SEAWAT (Langevin, et al., 2007), PEST (Doherty, 2016), BeoPEST (Hunt, et al., 2010), PESTPP-IES (Welter, et al., 2015; White, 2018; White, et al., 2020), Zone Budget (Harbaugh, 1990), and GSFLOW (Regan, et al., 2018). The latter couples PRMS (Markstrom, et al., 2015) and MODFLOW-NWT for integrated surface water and groundwater modeling.


This text focuses on the use of PM and does not delve into modeling theories or the functionality of the various packages. For more detailed information, users are encouraged to consult the user guides for the supported model codes. Additionally, several comprehensive textbooks on groundwater modeling are available. For example, Anderson, et al. (2015) provides an extensive reference on applied groundwater modeling, while Doherty (2015) offers an in-depth exploration of calibration and uncertainty analysis.

The most efficient way to get familiar with PM is by working through the tutorial available at [this link](#). In addition to the tutorial, several example projects are available online. Download links for these examples can be found in the "Application and Data Folders" section on page vi of the document.

1.2. Main Window of PM

The main window of PM and its major features are shown in Figure 1. A summary of these features is provided below. You can drag and rearrange the toolbars, Settings window, and Properties window as needed.

① Menu Bar and Toolbars

- The Menu bar contains several menus, detailed in Chapter 3 . The notation *Menu > Item* is used to indicate menu navigation. For example, *File > Save* refers to the "Save" option within the "File" menu.
- The File menu allows you to save or open PM files and create or import models. A PM file, with the extension “.pmproj”, contains data for all loaded models and graphical objects, except for online base maps. The Discretization, Parameters, and Models menus apply only to the active model (see TOC below) and are enabled only when at least one model is loaded under the Models group in the TOC.
- Many model parameters must be specified during the modeling process. To edit a model parameter, select the corresponding menu item from the Discretization, Parameters, or Models menus. The model parameter being edited is referred to as the *current data*. To complete the editing process, select *Tools > Finish Edit* or click the corresponding  button on the toolbar.
- To provide a quick overview of the data status, most items in the Discretization, Parameters, and Models menus are accompanied by one of the following icons. A menu item will appear disabled and grayed out if it is not applicable or not required.

- ✓ Indicates that the model data has been specified and will be used in the simulation.
- Indicates that the model data specified but will not be used in the simulation.
- No Icon** Indicates that model data has not been specified or that the item is not associated with input data, such as *Models > MODFLOW > Run*.

- The toolbar buttons serve as shortcuts to various menu items. A summary of these buttons is provided in Table 1.
- You can right-click on the area containing the menu bar or toolbars to access a menu that allows you to toggle the visibility of toolbars, as well as the Settings and Properties windows.

② **Table of Contents Window (TOC)** – The TOC contains a list of predefined groups (e.g., Views, Captions, Image Overlays, Raster Maps, Base Maps, Shapefiles, and Models). Each group may include a list of items (e.g., maps, shapefiles, models, etc.), where each item represents a map layer containing a visual representation of the item’s data. When an item is clicked, it is highlighted and becomes the *active layer*. Only one layer can be the active layer at a time. The Settings and Properties windows (see below) display the settings and values for the active layer. The major functions of the TOC are outlined below (refer to the following chapters for more details).

- Right-click on a group to open a popup menu, allowing you to add, remove, show, or hide items.
- Right-click on the Models group to create or import models. When multiple models are loaded in the Models group, only one model can be set as the *Active Model*, and the Discretization, Parameters, and Models menus will only apply to the active model. The active model is highlighted in yellow in the TOC.
- Right-clicking on a model item displays a pop-up menu with the following menu items:
 - Set as Active Model¹ – set the model as the active model.
 - Zoom to Model – zoom in to the extent of the model.
 - Remove Model¹ – remove the model and delete its data.
- Right-clicking on a model’s package displays a popup menu with options to Activate Package, Deactivate Package, Remove Package, Export to Shapefile, and more.
- Right-click on an item to display a popup menu with item-specific functions, such as Zoom To, Export to Shapefile or Export Matrix Data.
- The checkbox next to a group determines whether its items are displayed or hidden.
- The checkbox next to an item determines whether it is displayed or hidden. The data of a hidden item remains in the computer’s memory. If the item is no longer needed, removing it is recommended.

¹ This option is disabled when the Grid Editor (Section 3.4.1) or the Data Editor (Section 3.4.2) is being used.

③ **Settings Window** – This window displays the available visualization settings of the active layer. If a setting contains a text string, double-click on the text string to edit it.

④ **Map Viewer** – This displays base maps, shapefiles, model grid, model cell features such as wells or stream, and visual representations of model results, such as flow vectors, path lines, head contours, and more.

- You can interact with map using the Pan tool, the Zoom tool, and other tools described in Section 3.6.
- Many map layers feature on-map legends that can be enabled by checking the Visible checkbox in the Settings window of the respective map layer. The captions and contents of the on-map legends are synchronized with the TOC, with the exception that each individual on-map legend can display a maximum of 40 items. On-map legends can be moved or resized using the mouse. Their positions are anchored to the lower-left corner of the Map Viewer, making them independent of the map's zoom level and content position.
- The Map Viewer is not a full-fledged GIS system. Rather it is designed to display model input data and results, as well as to facilitate the data exchange between models and other GIS and online map systems.

⑤ **Properties Window**

- If the active layer is a shapefile, this window displays the properties of the selected feature within the shapefile.
- If the active layer is a model or one of its items (such as model grid), this window displays the meta data of the model and the aquifer parameters of the cell pointed to by the *grid cursor*. You may click on the [...] button next to an aquifer parameter to take notes about that parameter, such as data source.
- If a model data, such as Specific Storage or Well Package, is being edited, that data is highlighted in yellow.









⑥ **Status bar** – This displays the following information

- The name of the model data set or package that is being edited. For example, *Specific Yield* or *Drain (DRN)*.
- Current stress period, if the package that is being edited is time dependent.
- Ground surface elevation² at the mouse cursor's location on the map. Elevation values are provided for reference purposes only and do not represent precisely measured or surveyed values. The ground surface elevation data is provided in the following manner.

² Ground surface elevation is displayed only if the Geographic coordinate system is used. See Section 3.5.12 for details about the available coordinate systems.






















- Within the United States, Canada, and Mexico, the elevation data is provided by the EPQS³ of the National Map or the OpenTopoData⁴. The services are based on the 10m National Elevation Dataset (Gesch, et al., 2002).
- Outside of the United States, the elevation data is provided by OpenTopoData⁴ based on the Advanced Spaceborne Thermal Emission and Reflection Radiometer (ASTER) global 30m DEM datasets, with coverage from -83 to 83 degrees latitude.
- Coordinates at the mouse cursor's location on the map. The contents shown depend on the selected Coordinate system (see Section 3.5.12 for details).
 - If the Geographic Coordinate System is used, coordinates are expressed in the World Geodetic System 1984 (WGS 84) datum (National Imagery and Mapping Agency, 1984), with the corresponding EPSG code of 4326. For geographic coordinates, negative values represent longitudinal coordinates in the western hemisphere and latitudinal coordinates in the southern hemisphere. In addition to geographic coordinates, the WGS 1984 World Mercator (EPSG code 3395) or Universal Transverse Mercator, UTM (Defense Mapping Agency, 1989) coordinates may also be displayed. The format for UTM display is "UTM zone Hemisphere: (x, y)," where "zone" refers to the UTM zone number (ranging from 1 to 60) and "Hemisphere" indicates either "N" for North or "S" for South. The coordinates are given as x and y.
 - If the Cartesian Coordinate System is used, the coordinates are displayed as x and y.
- Row, Column, Layer of the active model at the mouse cursor location.










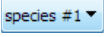
Table 1. The tool bar buttons

Button	Description
	Start a new PM project in the default coordinate system (see Section 3.5.1 for details).
	Open an existing PM project from a previously saved PM file with the ".PMPROJ" extension.
	Save the currently opened PM file. For a new project, you will be prompted to select a new PM file.
	Save the map as an image file.
	Open an Animation dialog box for creating animation files based on model results.
	Pan Tool. Use this tool to pan, zoom, and move the graphics in the map viewer.
	Zoom Tool. Use this tool to zoom in to an area by dragging a rectangular window on the map viewer.
	Observation Tool. Define observation points by clicking on the map.

³ <https://apps.nationalmap.gov/epqs/>

⁴ <https://www.opentopodata.org>

Button	Description
	MODPATH Tool. Define particle starting points by clicking on the map. Drags a rectangular window to define multiple starting points within the area. This button is active only when the active layer is a MODPATH instance.
	Measure Tool. Use this tool to measure the distance between two points or along a series of points.
	Row Cross Section Tool. Use this tool to visualize model cross sections and edit model data in the row cross section view.
	Column Cross Section Tool. Use this tool to visualize model cross sections and edit model data in the column cross section view.
	Pin Tool. Push down this button to disable map pan and zoom with the mouse.
	Rotation Tool. Push down this button to orient the display of the active model grid horizontally.
	Location Tool. Click this button to open the Locator dialog box that allows you to zoom to a cell or a point.
	Finish editing the current data.
	Remove the highlight. In the data editor (Section 3.4.2), double-clicking a cell highlights all other cells with the same value. The highlight mask can be set to clear automatically (see Preferences in Section 3.5.12). Click this button to manually clear the highlight mask if it hasn't been cleared automatically.
	Switch to the cell-by-cell input method.
	Switch to the polyline input method.
	Switch to the polygon input method.
	Switch to the Pilot Points input method.
	Toggle the <i>Copy Cell Data</i> function on or off. When enabled, the current cell's value(s) will be copied to all cells traversed by the grid cursor.
	Toggle the <i>Copy Layer Data</i> function on or off. When enabled, moving to another layer will copy the current layer's cell values, polylines, and polygons to the destination layer.
	Toggle the <i>Copy Grid Size</i> function on or off. When enabled, the current cell's column and row width will be copied to all cells traversed by the grid cursor.
	Toggle the <i>Copy Grid Split Factor</i> function on or off. When enabled, the current cell's column and row split factors will be copied to all cells traversed by the grid cursor.
	Set the temporal discretization parameters using the Temporal Discretization dialog box.
	Manage the stress period data with the Stress Period Data dialog box.
	Navigate to the previous stress period or select a specific period from the dropdown menu next to the  button.

Button	Description
	Navigate to the next stress period or select a specific period from the dropdown menu next to this button.
	Navigate to the previous time step or select a specific time step from the dropdown menu next to the  button.
	Navigate to the next time step or select a specific time step from the dropdown menu next to this button.
	Define the layer properties with the Layer Property dialog box.
	Move to the layer above or select a specific layer from the dropdown menu next to the  button.
	Move to the layer below or select a specific layer from the dropdown menu next to this button.
	Open the Transport Settings dialog box to define species and choose transport packages.
	Select a species to edit its data.

1.3. Model Settings

You can create or import a model by *File > Create Model* or *File > Import Model*. Once a model is created or imported, a corresponding model item will be added to the TOC. By clicking on the model item on the TOC, you have access to the model's Settings window (Figure 2) that includes the following functions.

1. General

- **Name:** This text string is the name of the model displayed on the TOC.
- **Model ID:** This text string serves as the unique identifier for the model and is used as the name of the *model data folder* where generated model input/output files are stored. The model data folder is located within the folder containing the PM file. If the Model ID is changed, PM will prompt you to confirm whether you want to rename the model data folder. To avoid compatibility issues with model codes, it is recommended to use only ASCII characters for the Model ID.
- **Notes:** This is for writing notes about the model. Click the [...] button to open the Model Notes dialog and edit the notes.
- **Wrap Arrays:** This setting controls how model data arrays are written to the model input files. For example, if Wrap arrays every 20 columns is selected, each row of a data array will contain a maximum of 20 values, and the remaining values will be written to the next line. This process continues until all values of the array are written. Arrays in all template files for PEST are always written in a wrapped form, even if "Do

not wrap arrays” is selected. In that case, arrays in template files are wrapped every 100 columns.

- **Highlight Parameter:** When editing an aquifer parameter, you can double-click on a cell (or press the Insert key) to highlight all other cells with the same value as the clicked cell. You can select a parameter from the dropdown box, and the values of the selected parameter will be used for highlighting.

2. Space and Time

- **Georeference⁵:** This contains the parameters used to associate the model grid with a spatial location. There are two methods for defining the location of the model grid. Keep in mind that the user-specified widths of model rows and columns are used in the simulation, even if the display of the model grid is stretched.
 - Click the *Edit* button to open the Georeference dialog box (Figure 3) where you can edit the geo-referencing data. **Georeferencing Method** defines how the model grid is associated with a spatial location.
 - **Four Grid Corners:** If this method is selected, the user-specified coordinates of the four model grid corners are used to position the model grid on the map.
 - **Lower-Left Grid Corner and Rotation:** If this method is selected, the grid azimuth and the coordinates of the lower-left grid corner are used to calculate the coordinates of the remaining grid corners. These calculated coordinates are then used to position the model grid on the map. The Grid Azimuth is the angle in degrees measured clockwise from a meridian to the axis along the model's row direction.
 - Check the checkbox (next to the *Edit* button) to display a bounding box of the model grid, along with four vertices at the corners of the bounding box. To move (translate) the model grid, hold the Shift-key while dragging one of the vertices.

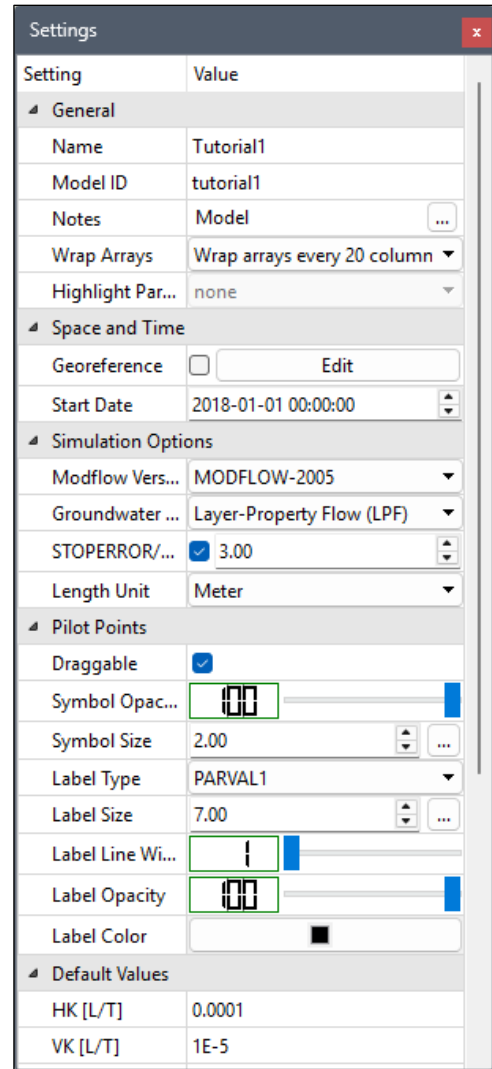


Figure 2. The Settings Window of a model

⁵ You may subsequently move and/or rotate the model grid with the Pan Tool (Section 3.6.1). **The georeferencing parameters only affect the location and appearance of the model grid on the Map Viewer, i.e., they do not affect the widths of model columns/rows used in the simulation.**

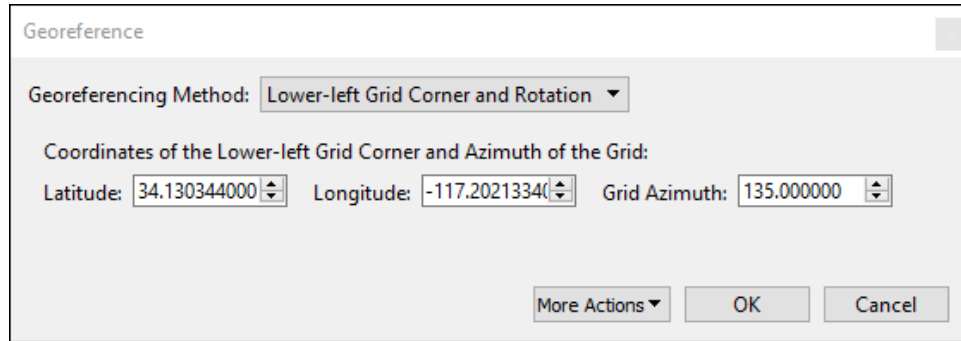


Figure 3. The Georeference dialog box

You can drag the vertices to the desired locations. Uncheck the checkbox to hide the bounding box and its vertices.

- **Start Date** is the start date/time of the flow simulation.

3. Simulation Options

- **MODFLOW Version:** Three MODFLOW versions are supported and installed by PM: MODFLOW-2005 (Harbaugh, 2005), MODFLOW-NWT (Niswonger, et al., 2011), and MODFLOW-OWHM (Boyce, et al., 2020). The groundwater flow simulation will be carried out using the selected MODFLOW version. The choice of MODFLOW version influences the availability of the groundwater flow package (see below). The combination of the selected MODFLOW version and groundwater flow package determines the availability of solver packages, which can be accessed under the Models > Modflow > Solvers menu (see Section 3.9.1.2 for details). PM enables or disables corresponding solver packages in the menu based on their availability.

MODFLOW-NWT should be selected if you plan to use GSFLOW.

- **Groundwater Flow Package:** The groundwater flow packages define aquifer properties, such as hydraulic conductivity or transmissivity. Three groundwater flow packages are supported by PM: the Block-Centered Flow (BCF) package, the Layer Property Flow (LPF) package, and the Upstream Weighting (UPW) package. The UPW package is only available in MODFLOW-NWT or MODFLOW-OWHM. One and only one groundwater flow package must be used in all models, and you can switch between supported packages at any time.

The UPW package is designed to address problems involving drying and rewetting nonlinearities of the unconfined groundwater-flow equation. The LPF and BCF packages share the same conceptualization, with differences mainly in the required input data. For details about the input data required by each groundwater flow package, refer to Section 3.7.3.

The UPW Package is recommended if you plan to use GSFLOW.

- **STOPERROR/STOPER:** This setting applies to the BCF and LPF packages. Check this box to allow the model to proceed to the next time step if it failed to converge and the budget percent discrepancy is below the specified value. If the UPW package is

selected as the groundwater flow package, use the “CONTINUE” option in the Newton Solver (Section 3.9.1.2.6) package instead.

- **Length Unit** is the length unit of all model data values that involve length.
4. **Pilot Points:** This group controls the appearance of pilot points that can be added to the model aquifer parameters by using the Pilot Points Input Method (Section 3.4.2.4).
- **Draggable:** This box determines whether pilot points can be dragged using the mouse.
 - **Symbol Opacity:** This controls the opacity of pilot point symbols. The symbols are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0. The colors of the pilot point symbols are determined by the PARTRANS values of pilot points:
 - Black if PARTRANS = none
 - Green if PARTRANS = log-transformed
 - Red if PARTRANS = fixed
 - Blue if PARTRANS = tied
 - **Symbol Size:** This is the size of pilot point symbols, expressed in the model length unit.
 - **Label Type:** The labels are drawn based on the selected data associated with pilot points. The data of pilot points are used in conjunction with Kriging and/or PEST. For more details, refer to Sections 3.4.2.4 and 3.9.5.2.
 - **Label Size:** This is the height of the label text, expressed in the model length unit.
 - **Label Line Width:** This is the line width (in pixels) used to draw the labels.
 - **Label Opacity:** The labels are opaque if the value of opacity is 100, translucent if the value is between 1 and 99, and transparent if the value is 0.
 - **Label Color:** Use this dropdown box to select a color for the labels.
5. **Default Values:** If a required aquifer parameter is not specified, its default value will be used in the simulation.

Chapter 2. Maps

This chapter describes the mapping features of PM, which are designed to enable modelers to align models with real-world locations and utilize existing raster or vector data from Geographic Information Systems (GIS). However, PM is not a GIS and does not offer advanced GIS editing features.

2.1. Views

As shown in Figure 4, a view consists of one or more polygons and serves as a navigation tool for specific geographical regions. To use Views, follow the instructions below.

1. To Add a View:

- Right-click on the Views group in the TOC and select **Add Current View** to add the current map area as a View.
- If a polygon shapefile is displayed, right-click on a polygon feature to open a popup menu, then select **Create View** to generate a View based on the selected polygon feature. If the polygon feature contains multiple rings (e.g., outer and inner rings, where inner rings represent holes within the outer ring), the View will include multiple polygons to accurately represent those rings.

2. To Remove Views:

- To remove a specific View, right-click on its name in the TOC to open the popup menu and select **Remove Item**.
- To remove all Views, right-click on the **Views** group in the TOC, open the popup menu, and select **Remove All Items**.

3. To Work with a View:

- To zoom to a View, right-click on its name in the TOC to open the popup menu and select **Zoom to**.
- To export a View as a shapefile, right-click on its name in the TOC to open the popup menu and select **Export to Shapefile**.
- To modify the shape of a View, drag its vertices to the desired locations.

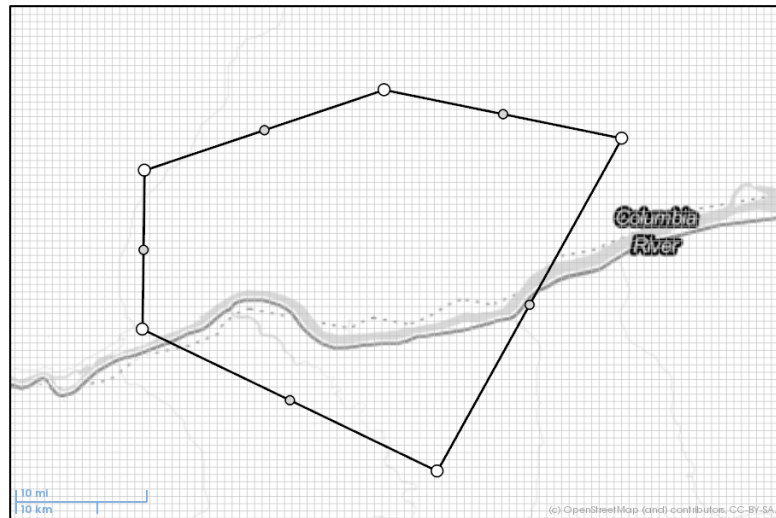


Figure 4. A View shown on the map viewer

- To move a View to another location, hold the Shift key while dragging one of its vertices.
 - A View has two types of vertices, distinguished by size and color:
 - Dragging a small gray vertex transforms it into a large white vertex, adding two new small gray vertices next to it.
 - To remove a large white vertex (along with its neighboring small vertices), right-click on it and select **Remove Vertex** from the popup menu.
 - If a View contains multiple polygons, you can remove a polygon by right-clicking on one of its vertices and selecting **Remove Polygon**.
- 4. To Customize the Display of a View:** Click on the View's name in the TOC to display its **Settings** window, where you can adjust the following options:
- **Name:** The name of the View displayed in the TOC. Double-click the text to modify it.
 - **Opacity:**
 - 100 – Fully opaque
 - 1–99 – Translucent
 - 0 – Fully transparent
 - **Polygon Color:** Use the dropdown menu to select the View's color.
 - **Show Vertices:** Check or uncheck this box to toggle the display of vertices. Turning off vertex display can help prevent unintentional modifications.

2.2. Captions

Captions are text strings that can be added to the Map Viewer. Captions are anchored to the relative position of the Map Viewer window and do not move with the maps.

1. To Add a Caption:

- Right-click on the Captions group in the TOC and select Add Caption.

2. To Remove Captions:

- To remove a specific caption, right-click on its name in the TOC to open the popup menu and select **Remove Item**.
- To remove all captions, right-click on the Views group in the TOC to open the popup menu and select **Remove All Items**.

3. To Work with a Caption:

- Move a caption by dragging it.
- Resize a caption by dragging its corners or borders.

4. To Customize the Display of a Caption:

Click on the caption's name in the TOC to open its **Settings** window, where you can modify the following options:

- **Name:** The name of the caption displayed in the TOC. Double-click to modify
- **Caption Text:** The actual text of the caption. Double-click to edit. If an active model is present, you can insert the following codes into the text string. These codes will be

dynamically replaced with the current date or layer of the active model when switching across time or layers:

- `\n` – line separator.
- `{name}` – The name of the model dataset or package being edited.
- `{yyyy}` – Year. `{m}` – Month. `{d}` – Day.
- `{hh}` – Hour. `{mm}` – Minute. `{ss}` – Second.
- `{sm}` = Short month name. `{lm}` = Long month name.
- `{p}` = Stress period. `{t}` = Time step. `{l}` = Layer
- **Enable Interaction:** Check this box to allow moving and resizing the caption with the mouse.
- **Opacity:**
 - 100 – Fully opaque
 - 1–99 – Translucent
 - 0 – Fully transparent
- **Color:** Use the dropdown menu to select the caption color.
- **Font Family, Font Weight, and Font Style** – Define the font appearance.
- **Text Alignment** – Use the dropdown menu to adjust text alignment.
- **Text Shadow:** Check this box to add a shadow effect to the caption.

2.3. Image Overlays

Image files, such as logos, can be added as **image overlays** in the **Map Viewer**. An image overlay is anchored to the relative position of the **Map Viewer** window and does not move with the maps.

1. To Add an Image Overlay:

- Right-click on the **Image Overlays** group in the TOC, select **Add**, and then choose an image file from the **Open File** dialog box.

2. To Remove Image Overlays:

- To remove a specific image, right-click on its name in the TOC to open the popup menu and select **Remove Item**.
- To remove all images, right-click on the **Image Overlays** group in the TOC to open the popup menu and select **Remove All Items**.

3. To Work with an Image Overlay:

- Move an image by dragging it.
- Resize an image by dragging its corners or borders.

4. To Customize the Display of an Image Overlay:

Click on the image overlay's name in the TOC to display its **Settings** window, where you can modify the following options:

- **Name:** The name of the image overlay displayed in the TOC. Double-click to modify.
- **Source File:** Displays the full path of the image file for reference. Modifying this text string will not affect the loaded image.

- **Notes:** A section for adding notes about this item. Click the [...] button to open the **Notes** dialog and edit or add notes.
- **Enable Interaction:** Check this box to allow moving and resizing the image with the mouse.
- **Opacity:**
 - 100 – Fully opaque
 - 1–99 – Translucent
 - 0 – Fully transparent

2.4. Raster Maps

Image files, such as digital aerial photos, satellite images, or scanned maps, can be geo-referenced and displayed as raster maps (i.e., offline base maps) in the Map Viewer. Additionally, images saved using File > Save Image (Section 3.5.9) can also be added as raster maps.

As shown in Figure 5, a raster map consists of an image and a polygon with eight vertices along its perimeter.

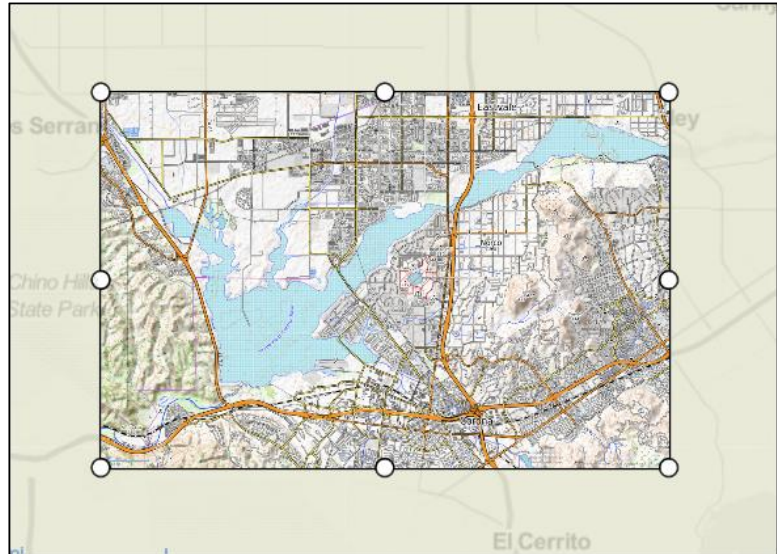


Figure 5. A raster map with its vertices

1. To Add a Raster Map:

- Right-click on the Raster Maps group in the TOC and select **Add** from the popup menu to display the Open File dialog box.
- In the Open File dialog box, select the desired image file and click Open to add it to the Map Viewer⁶. If the image size exceeds the Raster Map Image Size defined in Preferences dialog box (Figure 37), the image will be scaled down to the defined size.
- PM supports image files in the following formats:
 - Windows Bitmap (.bmp),
 - JPEG (.jpg),
 - Portable Network Graphics (.png),
 - Tagged Image File Format (.tif)

⁶ The contents of the added image are compressed and then embedded in the saved PM (.pmproj) files. This allows you to share .pmproj files with other users without having to send out image files separately.

If the optional projection and world files^{7,8} are available, PM uses their data to georeference the raster map. The extensions for these world files are:

- .pgw for PNG files
- .jgw for JPEG files
- .bpw for BMP files
- .tfw for TIFF files

2. To Work with a Raster Map:

- To zoom to a raster map, right-click on its name in the TOC to open the popup menu, then select **Zoom to**.
- To move (translate) a raster map to another location, hold the Shift key while dragging one of its vertices.
- To modify the shape of a raster map, drag its vertices to the desired locations. If you know the coordinates of the vertices on the four corners of the raster map, consider using **Georeference** (see below).

3. To Customize the Display of a Raster Map:

Click on its name in the **TOC**, then use the **Settings** window (Figure 6) as described below.

- **Name:** The text string represents the name of the Raster Map displayed in the TOC. Double-click the text string to modify it.
- **Source File:** The text string displays the full path of the image file and is for reference only. Modifying this string will not affect the loaded image.
- **Notes:** This field is for adding notes about the raster map. Click the [...] button to open the Notes dialog box for adding or editing notes.
- **Display Layer:** This is the layer number of the raster map in the Map Viewer. Items with smaller layer numbers are displayed on top of those with larger numbers.
- **Opacity:** Defines the opacity of a raster map.
 - 100 – Fully opaque
 - 1–99 – Translucent
 - 0 – Fully transparent
- **Polygon Color:** Use the dropdown box to define the color of the raster map's border.

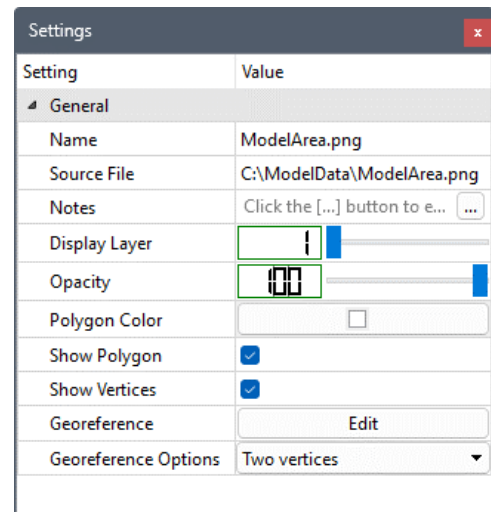


Figure 6. The Settings window for Raster Map

⁷ A projection file, usually with the file extension “.prj”, is a plain text file containing the geospatial reference information of the data stored in the “main” data file. A prj file has the same base filename as the “main” data file.

⁸ World files establish an image-to-world transformation that converts the image coordinates to real-world coordinates. See <https://www.loc.gov/preservation/digital/formats/fdd/fdd000287.shtml> for details.

- **Show Polygon:** Check/uncheck this box to toggle the display of the raster map's polygon.
- **Show Vertices:** Check/uncheck this box to toggle the display of the raster map's vertices. You may turn off the display to prevent accidental modifications.
- **Georeference:** Click **Edit** to open the **Raster Map Georeference** dialog box, where you can edit the coordinates of the four corners of the raster map. These corner coordinates are linearly interpolated to define the remaining vertices.
- **Georeference Options:** Use this dropdown to select the number of vertices for georeferencing:
 - Four vertices or Eight vertices: Allows you to drag the vertices to any position, deforming the map.
 - Two vertices (default): You can drag the displayed vertices, but the map's shape remains rectangular.

2.5. Base Maps

PM supports online maps from various raster tiled-map service providers. Several default map services are preprogrammed for convenience, including different styles offered by OpenStreetMap (<https://www.openstreetmap.org/about>). Additionally, users can add custom tiled-map services to PM.

⚠ Important: Always review the terms of use and limitations of a tiled-map service provider before adding it to PM.

The following sections outline the procedures for controlling the display of base maps in PM.

2.5.1. Edit Base Maps

To edit the base maps, right-click on the Base Maps group and select **Edit Data...** from the popup menu to open the Base Map dialog box (Figure 7). The features of this dialog box are described below.

1. **Table:** The table displays the parameters of tiled-map services loaded in the current session. Each row corresponds to a tiled-map service. The columns include:
 - : Click the checkbox repeatedly to switch between (checked), (partially checked), and (unchecked) states.
 - Checked: The map item will be added to the TOC and checked.
 - Partially Checked: The map item will be added to the TOC but remain unchecked.
 - Unchecked: If the corresponding map item exists in the TOC, it is removed.
 - **Map Name:** The name of the map displayed in the TOC.
 - **URL:** Defines the web address of a tiled-map service and its schema for accessing map tiles, such as:
`https://{s}.tile.openstreetmap.org/{z}/{x}/{y}.png,`

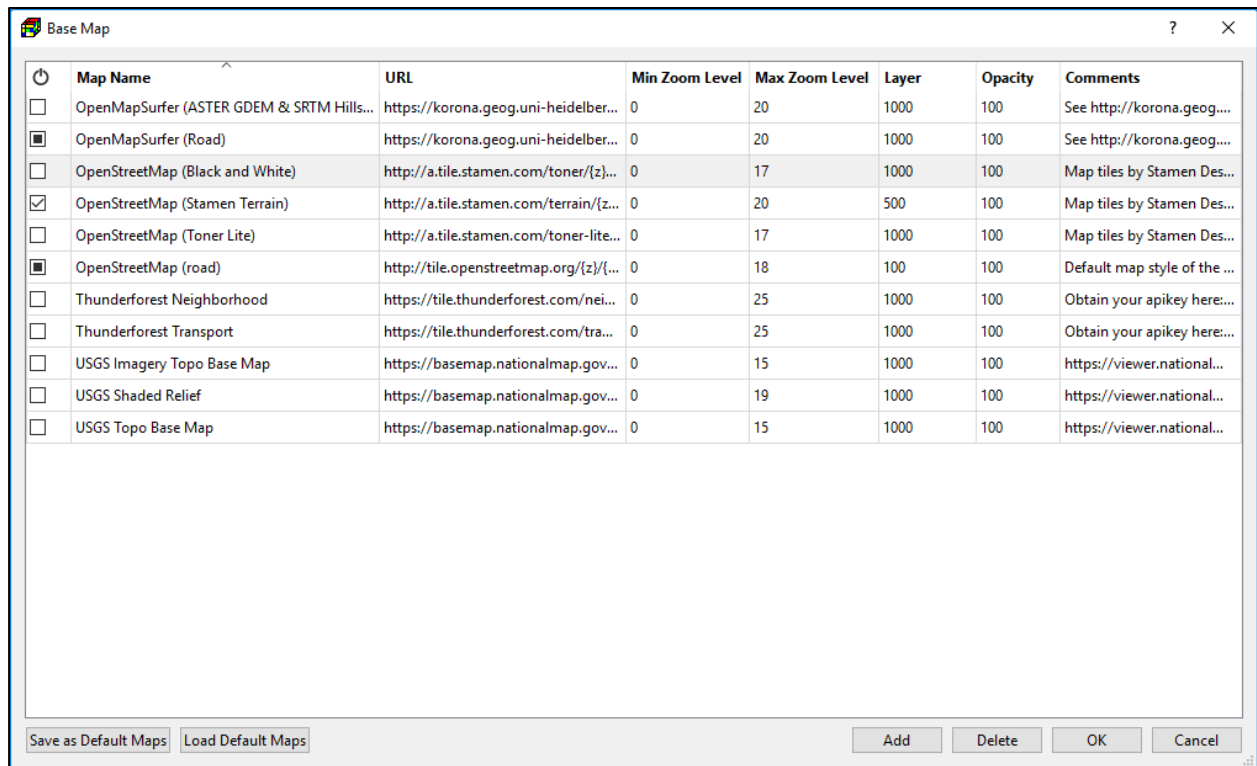


Figure 7. The Base Map dialog box

where {x}, {y} are placeholders for the tile coordinates, {z} represents the zoom level, and {s} is randomly replaced by a, b, c, or d. Some services use a quadkey (Schwartz, 2022) instead of {x}, {y}, {z} to uniquely identify a tile. In such case, use {k} as the placeholder for the quadkey.

A list of tiled-map services and URLs is available at: <https://leaflet-extras.github.io/leaflet-providers/preview/>

- Some services, such as Mapbox⁹ or MapTiler¹⁰, require additional parameters. You must add these parameters to the URL based on the service provider's requirements.
- For example, Mapbox's commercial Raster Tiles service requires an access token. To use it, register an account with Mapbox, create a free token, and replace your_token in the following URL:

`https://api.mapbox.com/v4/mapbox.satellite/{z}/{x}/{y}@.jpg90?access_token=your_token`

- **Min. Zoom Level/Max. Zoom Level:** Defines the zoom level range of a base map. Zoom level 0 covers the entire earth, with each subsequent level doubling precision in both horizontal and vertical dimensions. Most maps have a minimum zoom level of 0 and a maximum zoom level between 15 and 21..

⁹ www.mapbox.com

¹⁰ www.maptiler.com

- **Layer:** Determines the display order of base maps. Maps with smaller layer numbers are displayed on top of those with larger numbers.
- **Opacity:** Defines the opacity of a base map.
 - 100 – Fully opaque
 - 1–99 – Translucent
 - 0 – Fully transparent
- **Comments:** A field for additional information or credits to the service provider.

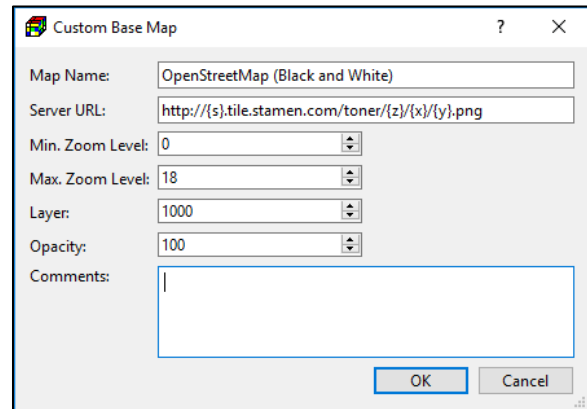


Figure 8. The Custom Base Map dialog box

2. **Add:** Click this button to open the Custom Base Map dialog box (Figure 8), which allows you to add a tiled-map service.
3. **Delete:** Click this button to remove the selected row from the table.
4. **Save as Default Maps:** Click this button to save the current table contents as the default map settings for new PM sessions.
5. **Load Default Maps:** Click this button to load the previously saved default map settings into the table.
6. **OK:** Click this button to close the Base Map dialog box. The base maps will be updated according to the table settings and displayed in the Map Viewer of the Main Window (Figure 1).

2.5.2. Customize the Display of a Base Map

When a base map is selected as the active layer, it can be modified through its Settings window. The fields in the Settings window are as follows.

1. General group

- **Name:** The text string represents the name of the map displayed on the TOC. You can double-click the text to modify it.
- **Display Layer:** This refers to the layer number of the base map in the Map Viewer. Items with smaller layer numbers are displayed on top of those with larger numbers
- **Opacity:** Defines the opacity of a base map.
 - 100 – Fully opaque
 - 1–99 – Translucent
 - 0 – Fully transparent
- **Min./Max. Zoom Level:** These parameters define the resolution range of the base map. Zoom level 0 represents the entire earth, and each subsequent zoom level increases the map's precision in both horizontal and vertical dimensions. Typically,

base maps have a minimum zoom level of 0 and a maximum zoom level between 15 and 21.

2.6. Shapefiles

The shapefile¹¹ format (ESRI, 1998) is a geospatial vector format used to describe spatial features, such as points, polylines, and polygons. Each spatial feature is associated with attributes, such as name or temperature. A shapefile consists of a collection of files with a common base filename and different file extensions, all stored in the same folder. Shapefiles can be displayed in the Map Viewer for reference purposes. The attributes of shapefile polygon features can be imported directly into model cells (See Section 3.6 for details).

When using a shapefile within PM, the following files are mandatory:

1. .shp – Contains the geometry of spatial features.
2. .dbf – Contains the attributes of each spatial feature.
3. .prj – Contains the coordinate system and projection information. This file, often referred to as a *spatial reference information file* or a *projection file*, is used only if the PM project utilizes the Geographic coordinate system (see Section 3.5.1 for details).
4. .shx – Contains an index of the spatial features, allowing fast forward and backward seeking by some software applications. While PM does not use this file, it will create it when exporting a shapefile.

2.6.1. Add Shapefiles to the Map Viewer

1. Right-click on the Shapefile group in the TOC, and select Add from the popup menu to display the Open File dialog box.
2. In the Open File dialog box, select the desired shapefile and click Open to add it to the Map Viewer.

Note: The added shapefiles are compressed and embedded in the saved PM (.pmproj) files. This enables you to share .pmproj files with others without needing to send the shapefiles separately.

2.6.2. Customize the Display of a Shapefile

By default, a shapefile is displayed in a single color. To customize the display, use its Settings window and Symbology dialog box, as described below.

2.6.2.1. The Settings Window

The following options are available in the Settings window for a shapefile.

1. General Group

- **Name:** The text string displayed on the TOC, used as the caption for the on-map legend. Double-click to modify.

¹¹ Follow these links [SURFER](#) or [AutoCAD](#) for details about exporting to shapefile from the respective software.

- **Source File:** The full path of the shapefile, provided for reference purposes only. Modifying this text does not affect the loaded shapefile.
- **Notes:** A place for adding notes about this item. Click the [...] button to open the Notes dialog box for adding or editing notes.
- **Identify By:** Select a field to identify similar features in the shapefile. When a feature is clicked, all features with the same value in the selected field will be highlighted.
- **Opacity:** Set the opacity value. The shapefile is opaque with an opacity value of 100, translucent between 1 and 99, and transparent with an opacity value of 0.
- **Display Layer:** This is the layer number of the shapefile in the Map Viewer. Items with smaller layer numbers are displayed on top of those with larger numbers.
- **Line Weight:** Defines the width of displayed line segments in pixels.
- **Colors:** Displays the current color(s) used. Colors can be defined using the **Symbology** dialog box. Click the [...] button to open the **Symbology** dialog box (discussed in the next section).
- **Pickable:** Check this box to make the shapefile pickable. This allows you to select shapefile features by clicking them.
- **Transformation:**

- When using the Geographic coordinate system (see Section 3.5.12) the Transformation option is deactivated as the shapefile uses its geospatial information data (proj file) for projection.
- When using the Cartesian coordinate system, the shapefile cannot be projected. However, you can transform the coordinates of all points in the shapefile by using the Transformation dialog box (Figure 9). The options in the dialog box are:

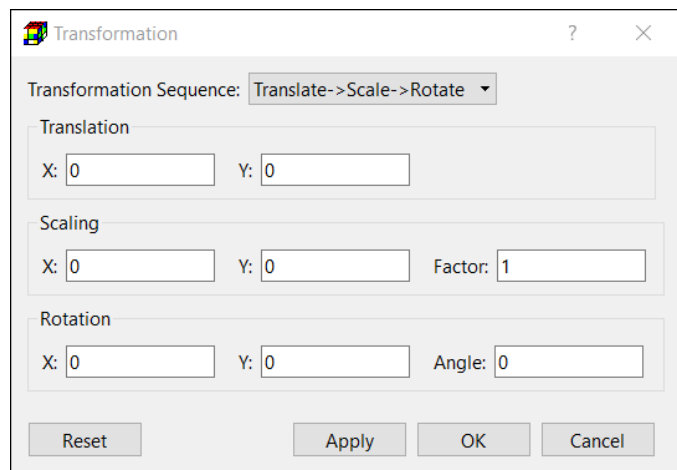


Figure 9. The Transformation dialog box

the shapefile by using the Transformation dialog box (Figure 9). The options in the dialog box are:

- **Transformation Sequence:** Defines the sequence of transformation operations. For example, Translate → Scale → Rotate will first translate the coordinates using the Translation group, then scale them using the Scaling group, and finally rotate them using the Rotation group.
- **Translation:** Moves all points in the shapefile by a set distance defined in this group:
 - X: Translation distance in the x- direction.
 - Y: Translation distance in the y-direction.

- **Scaling:** Scales the distance between the scaling center and all points in the shapefile by a scale factor:
 - X, Y: Coordinates of the scaling center.
 - Factor: The scale factor.
- **Rotation:** Rotates all points around a center by a given angle:
 - X, Y: Coordinates of the rotation center.
 - **Angle:** Rotation angle in degrees (positive for counterclockwise).
- **Reset:** Resets all transformation parameters to their default values.
- **Apply:** Applies the selected transformation sequence using the current transformation parameters.
- **OK:** Applies the selected transformation sequence and closes the dialog box.

2. On-map Legend Group

- **Visible:** Check this box to display the on-map legend.
- **Enable Interaction:** When checked, this option allows you to drag and resize the on-map legend.
- **Opacity:** The legend is opaque with an opacity value of 100, translucent when the opacity is between 1 and 99, and transparent when the opacity is 0.
- **Width:** Defines the relative width of the legend. A higher value results in a wider legend.
- **Reset Legend:** Click this button to reset the legend to its default size and position.

3. Label Group

- **Visible:** Check this box to display labels.
- **Field:** This dropdown contains the attribute fields of the selected shapefile. Use this option to choose the field for labeling the displayed shapes.
- **Display Label:** Controls when the labels should be shown.
- **Size:** Defines the font size of the labels.
- **Color:** Defines the color of the labels.

4. Point Feature Group:

(The settings in this group are ignored if point features are not present in the shapefile.)

- **Type:** Defines the shape of the point features.
- **Size:** Defines the appearance size of the point features.

5. Polyline Feature Group:

(The settings in this group are ignored if polyline features are not present in the shapefile.)


- **Label Distance:** Adjusts the distance between labels along the displayed polylines.

6. Polygon Feature Group:

(The settings in this group are ignored if polygon features are not present in the shapefile.)


- **Fill Polygon:** If checked, all polygon features of the loaded shapefile will be filled with the color(s) defined in the Symbology dialog box (see below).

2.6.2.2. The Symbology Dialog Box

The Symbology dialog box (Figure 10) allows modification of how attribute values are presented with spatial features in a shapefile. To open this dialog box, click the  button in the **Colors** field within the **Settings window** of the shapefile. Three color-mapping styles are available, selectable through the **Style** dropdown box:

1. Single Symbol (Figure 10):

This is the default style and the simplest way to change the color of a shapefile.

- **Color:** To choose a color, click the  button in the Symbol group, select a color from the Select Color dialog box, and click OK.
- **Label:** Enter a label or name for the shapefile and click OK. The label will be displayed in the shapefile legend.

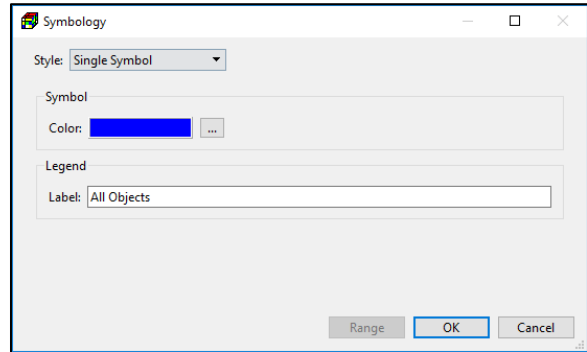


Figure 10. The Symbology dialog box - Single Symbol

2. Unique Values (Figure 11)

This style assigns a color to each unique value within a selected value field. For example, a shapefile with a land use field can display different colors for various land use types (e.g., residential, commercial, agricultural).

- **Fields:** Select a value field from the dropdown.
- **Color Scheme:** Choose a predefined color scheme.
- **Symbol:** This column shows the colors generated from the selected scheme. Double-click on a color to change it and choose a new one from the Select Color dialog box.
- **Value:** This column displays the unique values of the selected value field. These values cannot be modified.

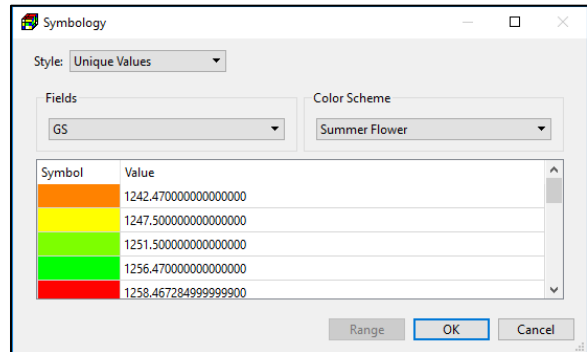


Figure 11. The Symbology dialog box - Unique Values

3. Graduated Colors (Figure 12)

This style groups quantitative values of a value field into classes, each identified by a distinct color. It's useful for displaying choropleth maps where quantitative data (e.g., concentration values) is mapped to a color ramp.

- **Fields Group:**

- **Value:** Select the field containing the desired quantitative values.
- **Normalization:** Choose a field to normalize the data in the **Value** field. PM divides the data in the **Value** field by the data in the **Normalization** field before mapping the results.

- **Classification Group:**

- **Classes:** Set the number of classes to display.
- **Color Ramp:** Select the color ramp to apply graduated colors to the classes.
- **Symbol:** This column displays the colors assigned to the individual classes. To change a color, double-click the color box and choose a new color from the Select Color dialog box.
- **Value:** This column shows the values assigned to the individual classes. To edit a value, click on it.
- **Range:** Click this button to specify the lower and upper bounds of the quantitative values to be mapped. The range is divided into equal intervals for each class.

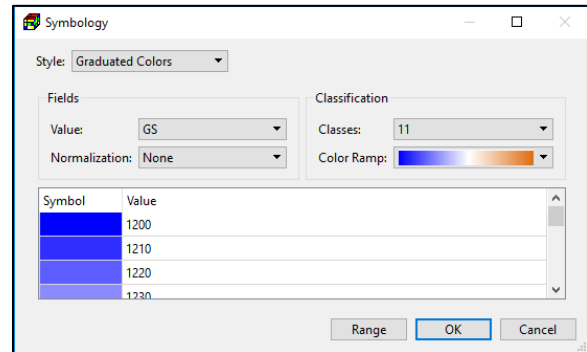


Figure 12. The Symbology dialog box - Graduated Colors

This Page Intentionally Left Blank

Chapter 3. Modeling Environment

This chapter serves as a complete reference for all menus and menu items of PM. The sections and subsections for the File, Tools, Discretization, Parameters, and Models menus are organized in the same order as they appear in the PM user interface. For example, the File menu is described in Section 3.5 (File Menu), and the Discretization menu is described in Section 3.7 (Discretization Menu).

3.1. Units

PM and its supported numerical models, except for GSFLOW, require the use of consistent units throughout the modeling process. For example, if you use meters as the unit of length [L] and seconds as the unit of time [T], hydraulic conductivity will be expressed in [m/s], pumping rates in [m³/s], and dispersivity in [m]. The simulation results will also follow these units. However, concentration values may sometimes have inconsistent units, such as using mg/Liter in a model where the length unit is meters. In this case, the calculated concentration will remain in mg/Liter, while the calculated mass (volume × concentration) will be expressed in [m³ · mg/Liter].

While GSFLOW allows the selection of independent units for precipitation, temperature, elevation, and stream discharge, most GSFLOW modules use predefined units for module-specific parameters. To avoid confusion, PM always displays the unit next to an input value.

Regardless of the language and regional settings of your computer, PM consistently uses a period (.) as the decimal point for decimal numerals.

3.2. Temporal Discretization

MODFLOW divides the total simulation time into discrete **stress periods**, which are further divided into **time steps**. After creating (see section 3.5.7) or importing (see section 3.5.8) a model, you can edit the stress periods and time steps under *Discretization > Time*. For more details, see Section 3.7.1.

The model input parameters are assumed to remain constant during a given stress period. For transient flow simulations that involve multiple stress periods, you can change parameters associated with most flow packages (e.g., River or Well) between stress periods. Similarly, for transport simulations, users can modify mass-loading rates or source concentrations for fluid sources and sinks across different stress periods.

PM uses the term **Simulation Time**¹² to refer to a specific time point in the simulation. It represents the elapsed time (in the consistent time unit as previously described) since the start

¹² Previous versions of PM used the terms “ElapsedTime” or “Elapsed Time”.

of a MODFLOW simulation. The simulation time begins at 0 and increases progressively throughout the course of the simulation.

3.3. Spatial Discretization

In a MODFLOW model, the aquifer system is represented by a discretized domain made up of an array of nodes and corresponding finite difference blocks (model cells). Figure 13 illustrates the spatial discretization of an aquifer system, depicting a rectilinear model grid composed of model cells with nodes located at the center of each cell, where hydraulic heads are calculated. Hydrostratigraphic units can be represented by one or more model layers, and the layer thickness may vary from cell to cell. After the hydraulic head values are computed, they are used to calculate the cell-by-cell flow terms, which are then used for water budget calculations, particle tracking, transport modeling, and visualization, such as flow vectors and contours.

PM uses the index notation [Layer, Row, Column] to describe the location of a cell within a 3D array. For instance, the cell located in the 3rd layer, 6th row, and 2nd column is written as [3, 6, 2]. For 2D arrays, the notation [Row, Column] is used. PM also assigns a unique Cell Number to each model cell, which is a sequential number starting from the first cell at row 1, column 1, and layer 1. Cell numbers increment first along columns, then rows, and finally layers.

The numbers of layers, columns, and rows as well as column and row widths are defined when creating a new model (Section 3.5.7) or importing an existing model (Section 3.5.8). These values can be modified using the Grid Editor. Model parameters (such as hydraulic conductivity or starting head) and features (such as wells or rivers) are assigned to individual cells using the Data Editor. Both the Grid Editor and Data Editor are described in the following section.

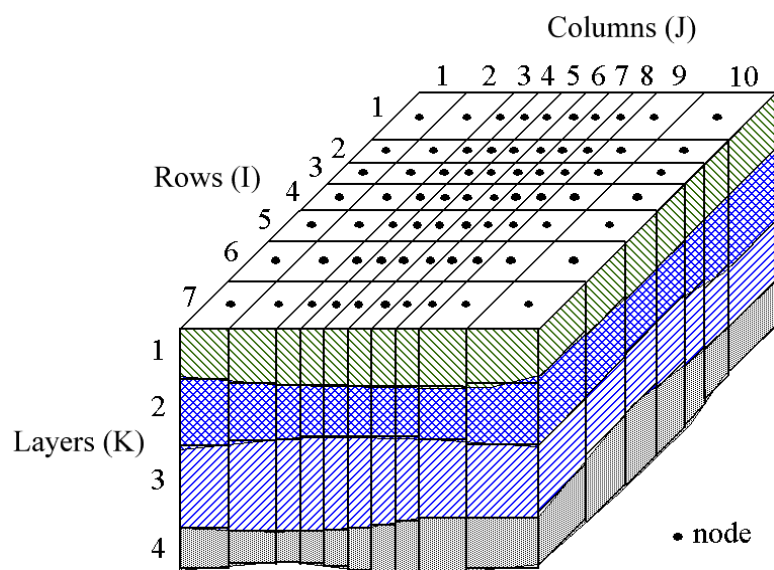


Figure 13. Spatial discretization of a Modflow Model

3.4. Grid Editor and Data Editor

3.4.1. Grid Editor

The Grid Editor is used to edit the finite difference model grid. You can adjust the column or row widths or refine the model grid by splitting columns and/or rows.

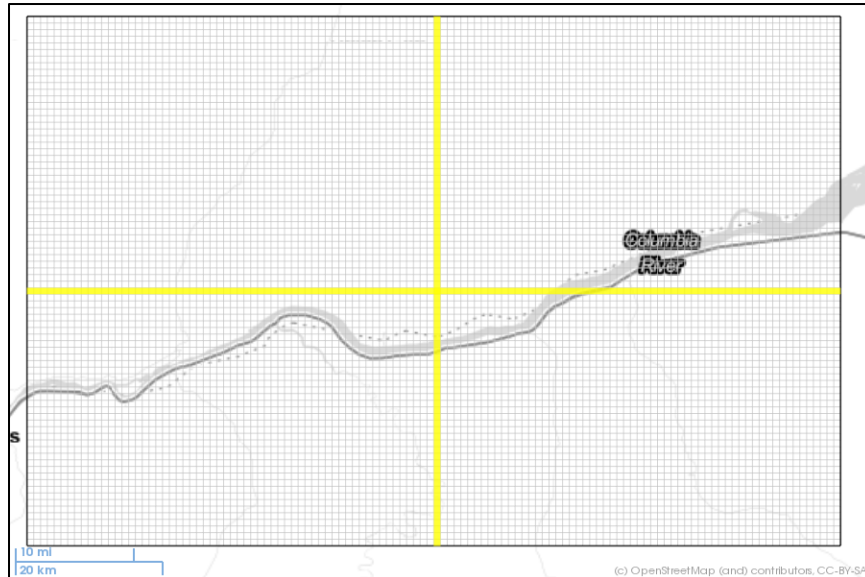


Figure 14. The Grid Editor with the grid cursor

- To Start the Grid Editor:** Select *Discretization > Grid*. Figure 14 shows the Grid Editor, with the grid cursor highlighting the entire row and column at the selected cell location. You can use the Pan tool (Section 3.6.1) and Zoom tool (Section 3.6.2) to interact with the map, to move the grid cursor, and/or adjust the model grid.
- To Modify the Column/Row Width and/or Refine the Grid:**
 - Right-click on the cell where the column/row width or refinement needs to be modified or use the arrow keys to move the grid cursor to the cell and press Enter. The Grid Size dialog box (Figure 15) will appear.
 - Modify the values in the Grid Size dialog box, then click OK to apply the changes.
 - Consider using the *Copy Grid Size* function (📏) on the toolbar and *Copy Grid Split Factor* function (📏) on the toolbar if you need to modify multiple rows/columns. For details about these functions, refer to Table 1 of Section 1.2.

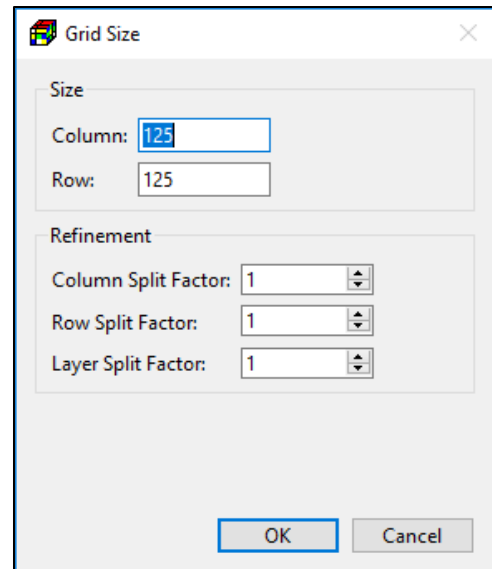


Figure 15. The Grid Size dialog box

3. To Shift or Rotate the Model Grid¹³:

- To shift the model grid using shift+drag: Place the mouse cursor over the grid cursor, press and hold the Shift key and the left mouse button, drag the model grid by moving the mouse, and then release the Shift key and left mouse button to place the model grid.
- To rotate the model grid using ctrl+drag: Place the mouse cursor over the grid cursor, press and hold the Ctrl key and the left mouse button, drag the mouse to rotate the model grid, and then release the Ctrl key and left mouse button to place the model grid. The center of rotation is the lower-left corner of the unrotated grid.

4. To Exit the Grid Editor: Click the button on the toolbar or select *Tools > Finish Edit*.

- If changes have been made in the Grid Editor, you will be prompted to save or discard the changes.
- If the grid has been refined, the model parameters are either maintained or scaled according to the new cell size, depending on their nature. The following rules apply.
 - Pumping rates of wells defined by WEL and MNW1, mass-loading rates (Section 3.9.3.3.4), and cell-by-cell conductance values for the river, drain, general-head boundary, and stream are scaled by the cell volume. For example, if a well cell is split into four cells, all four cells will be treated as wells, each with 1/4 of the original pumping rate. The sum of their pumping rates will remain the same as that of the previous single well.
 - Parameters of wells defined by MNW2 remain unchanged. However, an MNW2 well is moved to the split cell closest to its original location.
 - Transmissivity (T) and storage coefficient (S) values are scaled by the cell thickness.
 - The parameters of polylines and polygons remain unchanged, as they are grid independent. If streams (from the Stream (STR) or Streamflow-Routing (SFR2) package) were defined using the cell-by-cell input method, you must redefine the segment and reach numbers for the streams. For SFR2, the reach length must also be redefined. If streams were defined using polylines, you must re-apply the polylines to the model cells (see Section 3.4.2.2 for details on the Polyline input method).
 - If an unconfined layer is refined, only the first layer remains unconfined, while the others are set to convertible (i.e., capable of switching between confined and unconfined).
 - All other model parameters remain the unchanged.


¹³ You can also define the position of a model grid with the *Georeference* function in the Settings window of the model, see Figure 2.

3.4.2. Data Editor


The Data Editor is used to assign model parameters, such as HK, VK, drains, wells, etc., to individual model cells. It provides four input methods for most model parameters:

- The Cell-by-Cell input method allows direct assignment of values to model cells.
- The Polyline, Polygon, and Pilot Point input methods allow you to assign model parameters in other ways.
- The Pilot Point input method is available only when editing aquifer parameters, such as HK, VK, Ss, Sy, etc.


Regardless of the input method used, the simulation always utilizes the Cell-by-Cell model parameters. However, Pilot Points can be optionally used for pilot-point based parameter estimation with PEST or PESTPP-IES.

1. **To Start the Data Editor:** Select a menu item from the Discretization, Parameters, or Models menu. For example, select *Parameters > Horizontal Hydraulic Conductivity* to start the Data Editor for HK values.
2. **To Exit the Data Editor:** Click the Finish Edit button  on the toolbar or select *Tools > Finish Edit*.

3.4.2.1. Cell-by-Cell Input Method

To activate this input method, select *Tools > Cell-by-Cell Tool* or click the  button on the toolbar. You can move the *grid cursor* to a cell by dragging the grid cursor, left clicking on the cell¹⁴, or using the arrow keys. **Error! Reference source not found.** summaries the key assignments and their corresponding actions. The functions of the cell-by-cell input methods are as follows.

1. To Assign Values to a Cell:

- If the Pan tool (Section 3.6.1) is not activated, activate it by clicking on the  button on the toolbar or selecting *Tools > Pan Tool*.
- Right-click on the cell in the Map Viewer to open a dialog box for assigning cell data. Alternatively, move the grid cursor to the cell and press the Enter key to open the dialog box.
- In the dialog box, enter the new value(s) and click OK. For example, if you are editing HK values, you can enter an HK value and a Parameter/Zone Number in the Aquifer Property dialog box (Figure

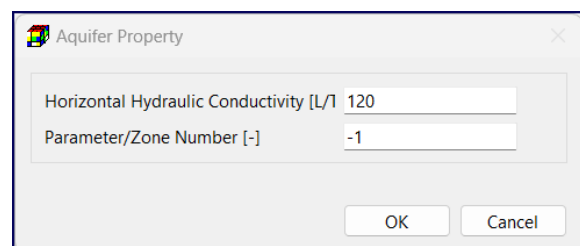




Figure 16. The Aquifer Property dialog box

¹⁴ If a pickable shapefile overlies the clicked cell, the shapefile will be picked instead of moving the grid cursor to the cell. To prevent this from happening, uncheck the Pickable box from the Settings window of the shapefile.

16). The *Parameter/Zone Number* is used in conjunction with PEST and/or Kriging. The following rules apply:

- If the *Parameter/Zone Number* is greater than zero: The specified value is used as a parameter number. PEST can be used to estimate a uniform parameter value for all cells with the same parameter number. Kriging does not apply to these cells.
- If the *Parameter/Zone Number* is less than zero: The absolute value of the specified number is used as a zone number. The Kriging method can be used to interpolate aquifer properties from pilot points to the cells within a specific zone. PEST can be applied to estimate aquifer properties at the pilot points. See Section 3.4.2.4 for details.
- If the *Parameter/Zone Number* is equal to zero: The specified aquifer property values will not be modified by PEST or Kriging.
- Some packages¹⁵ allow you to assign multiple instances of package-specific features to a cell. For example, you may assign multiple wells to a cell in the Well Package. In that case, the dialog box will include the following functions:
 - *More Actions > Add*: Add a new instance to the current cell.
 - *More Actions > Delete*: Delete current instance from the cell.
 - *More Actions > Delete All*: Delete all instances from the cell.
 - Click on   to go to previous or next instance within the current cell.


2. To Highlight Cells with the Same Value:

- In the Data Editor, double-click on a cell (or press the Insert key) to highlight all other cells with the same cell value as the clicked cell. The number of highlighted cells will briefly appear on the status bar. Since highlighting model cells may take some time, you can press the Esc key to interrupt the process.
- When editing an aquifer parameter, you can select a Highlight Parameter from the model's Settings window. The values of the selected Highlight Parameter will be used for highlighting.
- When editing a package, you can select a Highlight Parameter from the package's Settings window. The values of the selected Highlight Parameter will be used for highlighting.
- The highlight color with each repeated double-click.
- The highlight opacity may be modified by adjusting the value of *Highlight Opacity* in the Preferences dialog box.




3. To Copy Value(s) of One Cell to Another:

- Click the cell you want to copy.
- Click the  button on the toolbar. *Cell Copy* is enabled when the button is depressed.

¹⁵ The following packages allow multiple instances of package-specific features to be assigned to a cell: CHD, DRN, DRT1, FHB1, GHB, HFB6, MNW1, MNW2, RIV, STR, SFR2, and WEL.

- Move the grid cursor with the mouse or the arrow keys (←, →, ↑, ↓). The current cell's value(s) will be copied¹⁶ to the cells traversed by the grid cursor.
- Click the  button on the toolbar. *Cell Copy* is disabled when the button is raised. The button will automatically raise when moving to another layer or exiting the Data Editor.

4. To Copy Data (e.g., cell values, polylines, polygons) from One Layer to Another:

- Move the grid cursor to the layer you want to copy.
- Click the  button on the toolbar. *Layer Copy* is enabled when the button is depressed.
- Move the grid cursor to the target layer using the PgUp/PgDn keys or the Current Layer dropdown box next to the  button on the toolbar. The data from the layer will be copied to all layers traversed by the grid cursor.
- Click the  button on the toolbar. *Layer Copy* is disabled when the button is raised. The button will automatically raise when exiting the Data Editor.

5. To Reset, Modify, Import, or Export Values for the Current Layer or Entire Model: Refer to the *Tools > Data* menu (Section 3.6.20) for details.

3.4.2.2. Polyline Input Method¹⁷

To activate this input method, select *Tools > Polyline Tool* or click the corresponding button on the toolbar. If the Pan Tool (Section 3.6.1) is not already activated, enable it by clicking its button on the toolbar or selecting *Tools > Pan Tool*.

The usage of this input method involves the following straightforward steps:

1. Add polylines to the Map Viewer.
2. Assign data to one or more vertices of the polylines.
3. Apply the vertex data to model cells along the polylines using linear interpolation.

To assist with polyline editing, the Polyline Tool provides two pop-up menus:

- **Polylines Menu** (Figure 17): The Polylines Menu contains functions that apply to all polylines and can be accessed by right-clicking on the Map Viewer.
- **Polyline Data Menu** (Figure 20): The Polyline Data Menu contains functions specific to an individual polyline and can be accessed by right-clicking on a vertex of that polyline.

The functions of these menus are described below.

¹⁶ When editing MNW2, the wells of a cell will be added (instead of copied) to the cells traversed by the grid cursor.

¹⁷ The Polyline input method does not support all model parameters or packages. If a specific model parameter or package is not supported, this method is deactivated.

1. Polylines Menu (Figure 17)

- **Add Polyline:** Select this option to add a polyline to the map (Figure 18). The polyline is centered at the clicked location and extends to the east and west boundaries of the current view.
 - The polyline is associated with the current model layer and consists of three types of vertices, distinguished by size and color. Model parameter values can be assigned to the larger white vertices, including the one with a black dot at its center, which marks the polyline's starting point. When defining a stream with a polyline, ensure that the starting point is placed at the upstream end of the stream.
 - To edit vertex data, right-click on a white vertex and select Edit Vertex Data from the pop-up menu. A vertex with assigned data is marked with yellow brackets on the map. For more details, refer to the *Polyline Data Menu* section.
 - To modify the polyline's geometry, drag its vertices. Small gray vertices act as "pivot joints." When a small gray vertex is dragged, it transforms into a large white vertex, and two new small gray vertices are added adjacent to it.
 - To select a polyline, click on one of its vertices. A selected polyline is displayed in yellow, with its large white vertices displayed as ⊗.

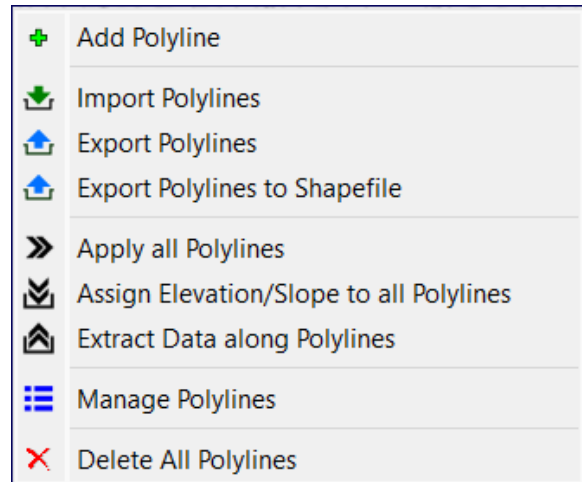


Figure 17. The Polylines Menu

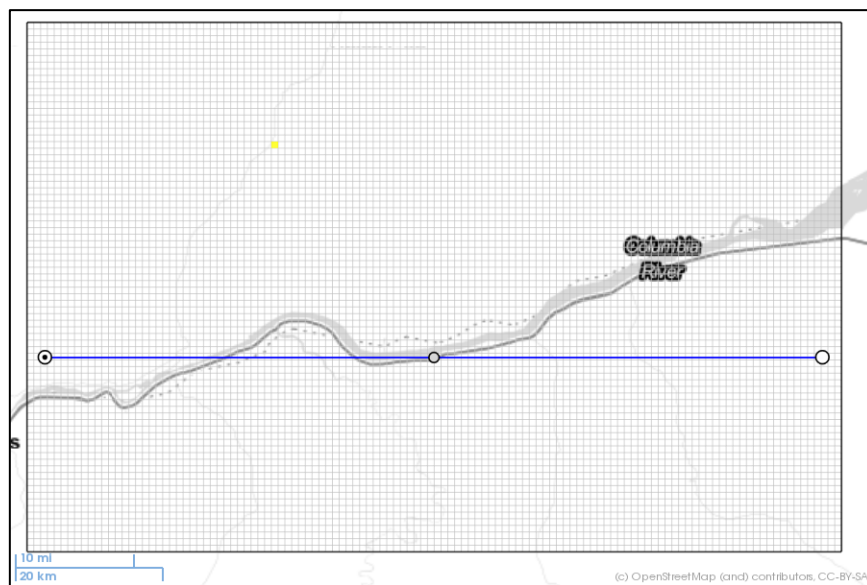


Figure 18. A polyline shown on the map viewer

- To move (translate) a polyline, use Shift + Drag:
 - Place the mouse cursor over a vertex.
 - Press and hold both the left mouse button and the Shift key.
 - Move the mouse to drag the polyline.
 - Release both the Shift key and the left mouse button to place the polyline in its new position.
- **Import Polylines:** Select this item to import polylines from a Polyline File (see Section 5.1.12 for the format) and add the imported polylines to the collection.
- **Export Polylines:** Select this item to export polylines to a Polyline File (see Section 5.1.12 for the format).
- **Export Polylines to Shapefile:** Select this item to save polylines in the shapefile format.
- **Apply all Polylines:** Select this option to apply the vertex data from all polylines to the corresponding model cells along their paths. The existing data in those model cells will be replaced with the vertex data. Vertex data are applied to model cells based on the following rules:
 - For cells located between two vertices with assigned data, the vertex data are linearly interpolated to those cells. The interpolation weights are determined by the distances along the polyline between the cell and the two vertices.
 - For cells not positioned between two data-assigned vertices, the data from the nearest vertex is assigned to the cell.
- **Append all Polylines:** Some packages¹⁵ allow multiple instances of package-specific features to be assigned to a single cell. For example, in the Drain Package, you can assign multiple drains to a single cell. Select this option to apply and append the vertex data from all polylines to the corresponding model cells along their paths. The vertex data are applied using the same rules as Apply All Polylines.
- **Assign Elevation/Slope to All Polylines** (Available when editing the SFR2 Package): Select this option to open the *Assign Elevation/Slope* dialog box, where you can choose GeoTIFF raster files and specify a multiplier to convert the extracted data to the desired unit. The dialog box extracts raster data from the selected GeoTIFF files and assigns the extracted values as the top elevation of the streambed (STRTOP) to the value-assigned vertices (marked with yellow brackets) of all polylines. The stream slope (SLOPE) at these vertices is then calculated based on the streambed elevation data and the distances between vertices.
 - If a valid raster value is unavailable for a vertex, its STRTOP and SLOPE values remain unchanged.
- **Extract Data along Polylines:** Select this option to open the Extract Data dialog box, which allows you to extract data from GeoTIFF raster files for points along the polyline. You will need to specify the following inputs:
 - Filenames of the GeoTIFF raster files
 - Filename for the output file

- Resolution (i.e., the distance between points along the polylines)
- Multiplier to convert the extracted data to the desired unit

The output file will contain a line of comma-separated values (CSV) for each point along the polylines, including:

- Polyline number and Polyline name (identifiers of the polyline)
- Latitude and Longitude (coordinates of each data point)
- Distance (running distance of the data point measured from the polyline’s starting point)
- Extracted Data

These elevation profiles can be used to derive 8-point cross-sections for the SFR2 Package.

- **Manage Polylines:** Select this item to manage properties of existing polylines using the Polylines dialog box (Figure 19). This dialog box contains a table where each row represents a polyline, with the following columns:

- **Polyline Visibility**

- Checked: The polyline is turned on and displayed in the *Map Viewer*. Its vertex data can be applied to model cells.
- Unchecked: The polyline is turned off, will not be displayed in the *Map Viewer*, and its vertex data may not be applied to model cells.

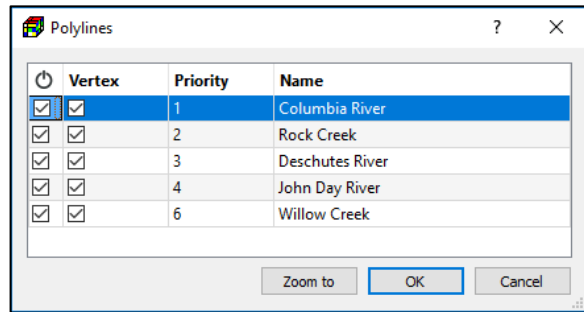


Figure 19. The Polylines dialog box

- **Vertex:**

- Checked: Vertices are displayed on the map.
- Unchecked: Vertices are hidden to avoid unintended modifications. Vertex data can still be applied to model cells even if vertices are not visible.

- **Priority:** Determines the order in which overlapping polylines are applied to cells. The data from polylines with lower priority numbers will overwrite the data from polylines with higher priority numbers.

- **Name:** Name of the polyline.

- **Delete All Polylines:** Select this item to delete all polylines.

2. **Polyline Data Menu** (Figure 20): This menu contains functions related to an existing polyline and can be accessed by right-clicking on a white vertex of that polyline. The available functions are as follows:

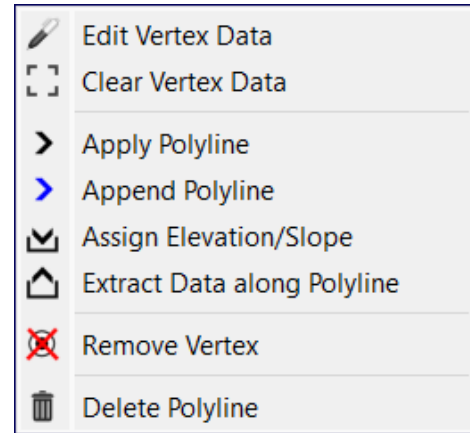


Figure 20. The Polyline Data Menu

- **Edit Vertex Data:** Select this option to open a dialog box for vertex data entry. Enter the necessary data and click OK to apply changes.
 - Vertices with assigned data are marked with yellow brackets on the map.
 - Most polyline data entry dialog boxes include **»»** buttons next to data entry fields. Figure 21 illustrates the Aquifer Parameter dialog box with these buttons. Clicking the button next to a parameter applies the data to the corresponding model cells along the polyline.
 - In packages¹⁵ that support multiple instances of package-specific features within a cell (e.g., the *Well* or *Drain* Package), these **»»** buttons only modify existing instances within those cells.

Figure 21 illustrates the Aquifer Parameter dialog box with these buttons. Clicking the button next to a parameter applies the data to the corresponding model cells along the polyline.

- **Clear Vertex Data:** Select this option to remove the specified data from a vertex.
- **Apply Polyline:** Use this option to apply vertex data from a polyline to model cells along its path. The existing data in those model cells will be replaced by the vertex data. The application follows these rules:

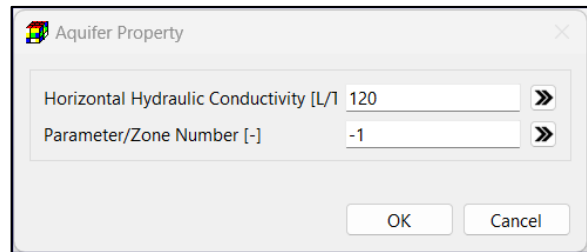


Figure 21. The Aquifer Parameter dialog box

- For cells located between two vertices with data, the vertex data is linearly interpolated based on their distances along the polyline.
- For cells not positioned between two data-assigned vertices, the nearest vertex's data is assigned to the cell.
- **Append Polyline:** Some packages¹⁵ allow multiple instances of package-specific features within a single cell (e.g., the *Well* or *Drain* Package). In such cases, selecting this option will append the current polyline's vertex data to the model cells along the polyline, instead of replacing existing data. The same application rules as **Apply Polyline** are followed.
- **Assign Elevation/Slope** (Available when editing the SFR2 Package): Selecting this option opens the Assign Elevation/Slope dialog box, allowing you to extract raster data from GeoTiff files and assign the extracted values as the top elevation of the streambed (STRTOP) for data-assigned vertices that are marked with yellow brackets. The stream slope (SLOPE) at these vertices is derived using the streambed elevation data and the distances between the data-assigned vertices.

- If a valid raster value is unavailable for a vertex, its STRTOP and SLOPE values remain unchanged.
- The following inputs must be specified in the dialog box:
 - Filenames of GeoTiff raster files
 - A multiplier to convert the extracted data to the desired unit
- **Extract Data along Polyline:** Select this option to open the Extract Data dialog box, which allows you to extract data from GeoTIFF raster files for points along the polyline. You will need to specify the following inputs:
 - Filenames of the GeoTIFF raster files
 - Filename for the output file
 - Resolution (i.e., the distance between points along the polylines)
 - Multiplier to convert the extracted data to the desired unit

The output file will contain a line of comma-separated values (CSV) for each point along the polylines, including:

- Polyline number and Polyline name (identifiers of the polyline)
- Latitude and Longitude (coordinates of each data point)
- Distance (running distance of the data point measured from the polyline's starting point)
- Extracted Data

These elevation profiles can be used to derive 8-point cross-sections for the SFR2 Package.

- **Remove Vertex:** Select this option to remove a large white vertex along with its neighboring small vertices.
- **Delete Polyline:** Select this option to delete the polyline.

3.4.2.3. Polygon Input Method¹⁸

To activate this input method, select *Tools > Polygon Tool* or click the corresponding button on the toolbar. If the Pan Tool (Section 3.6.1) is not already activated, activate it as well by clicking its button on the toolbar or selecting *Tools > Pan Tool*.

The usage of this input method involves the following straightforward steps:

1. Create polygons in the Map Viewer.
2. Assign data to the created polygons.
3. Apply this data to the model cells located within the polygons.

To facilitate polygon editing, the **Polygon Tool** provides two pop-up menus:

¹⁸ The Polygon input method does not support all model parameters or packages. If a specific model parameter or package is not supported, this method is deactivated.

- **Polygons Menu** (Figure 22): Contains functions related to all polygons and is accessible by right-clicking within the Map Viewer.
- **Polygon Data Menu** (Figure 25): Contains functions specific to an individual polygon and is accessible by right-clicking on a vertex of that polygon.

The functions provided by these menus are described in detail below.

1. Polygons Menu (Figure 22)

- **Add Polygon:** Select this option to add a polygon to the map (Figure 23). The polygon is associated with the current model layer and extends to the boundaries of the current view.
 - A polygon has two types of vertices, differentiated by size and color.
 - Model parameters can be assigned to a polygon by right-clicking on a white vertex and selecting Edit Polygon Data from the pop-up menu. For more details, refer to the *Polygon Data Menu* section.
 - To modify a polygon's geometry, drag its vertices. Small gray vertices act as "pivot joints." When a small gray vertex is dragged, it transforms into a large white vertex, and two new small gray vertices are added adjacent to it.
 - Select a polygon by clicking on one of its vertices. A selected polygon is highlighted with the Color for selected Polyline/Polygon defined in the Preferences dialog box.
 - Move (translate) a polygon using Shift + Drag:
 - Place the mouse cursor over a vertex.
 - Press and hold the Shift key and the left mouse button simultaneously.

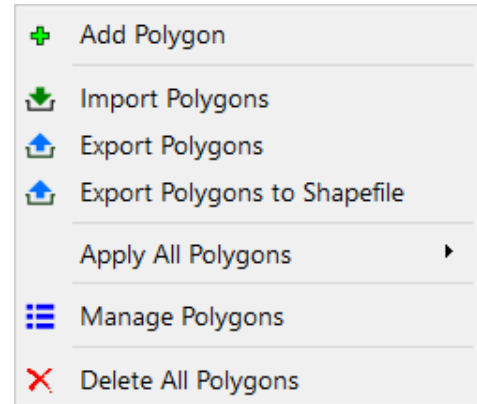


Figure 22. The Polygons Menu



Figure 23. A Polygon shown on the map viewer

- Move the mouse to drag the polygon.
- Release the Shift key and left mouse button to place the polygon.
- **Import Polygons:** Select this option to import polygons from a polygon file (Section 5.1.10).
- **Export Polygons:** Select this option to export polygons to a polygon file (Section 5.1.10).
- **Export Polygons to Shapefile:** Select this option to save polygons in shapefile format.
- **Apply All Polygons:** This option provides a submenu for applying polygon values to model cells located within the polygons. The resulting cell values depend on the selected method:
 - **Replace:** Final cell value = polygon value.
 - **Add:** Final cell value = current cell value + polygon value.
 - **Subtract:** Final cell value = current cell value - polygon value.
 - **Multiply:** Final cell value = current cell value × polygon value.
 - **Divide:** If polygon value ≠ 0, final cell value = current cell value ÷ polygon value. If polygon value = 0, the cell value remains unchanged.
 - **Minimum:** Final cell value = minimum(current cell value, polygon value)
 - **Maximum:** Final cell value = maximum(current cell value, polygon value)
 - **Average:** Final cell value = average(current cell value, polygon value)
- **Manage Polygons:** Select this item to manage properties of existing polygons using the *Polygons* dialog box (Figure 24). This dialog box contains a table where each row represents a polygon with the following columns:

- **⏻ Polygon Visibility.**

- Checked: Polygon is enabled and displayed in the Map Viewer. Its data can be applied to model cells.
- Unchecked: Polygon is disabled and hidden in the Map Viewer. Its data will not be applied to model cells.

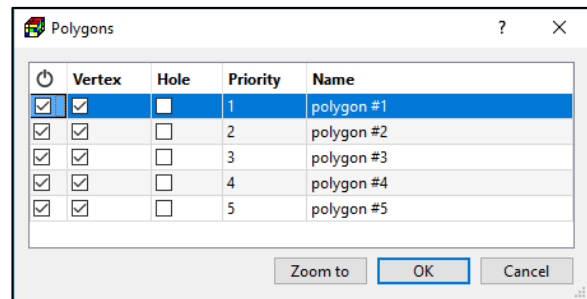


Figure 24. The Polygons dialog box

- **Hole:**

- Checked: The polygon is treated as a hole. Its data will not be applied to model cells.
- Unchecked: The polygon is not a hole. Its data can be applied to model cells.

- **Vertex:**

- Checked: Vertices are displayed on the map.
- Unchecked: Vertices are hidden to avoid unintended modifications. Polygon data can still be applied even if vertices are not visible.

- **Priority:** Determines the order in which overlapping polygons are applied to cells. The data from polygon with lower priority numbers will overwrite the data from polygon with higher priority numbers.
 - **Name:** The name of the polygon, used for identification purposes.
 - **Delete All Polygons:** Select this item to delete all polygons.
2. **Polygon Data menu** (Figure 25): This menu contains functions related to an existing polygon and can be opened by right-clicking on a white vertex of that polygon.
- **Edit Polygon Data:** Select this option to open a dialog box for polygon data entry. Enter the required data and click OK to apply the changes.
 - Most polygon data-entry dialog boxes include **»** buttons next to data entry fields. Figure 26, for example, shows the Drain Package dialog box with these buttons. Clicking the button next to a parameter replaces the data of cells located within the polygon with that parameter value.
 - In packages that support multiple instances of package-specific features within a cell (e.g., the Drain Package), these buttons only modify existing instances within the cells.
 - **Clear Polygon Data:** Select this option to remove specified data from the polygon.
 - **Apply Polygon Data**¹⁹ This submenu contains methods for applying polygon values to the cells located within the polygon. Final cell values are calculated according to the selected method:
 - **Replace:** Final cell value = polygon value.
 - **Add:** Final cell value = current cell value + polygon value.
 - **Subtract:** Final cell value = current cell value - polygon value.
 - **Multiply:** Final cell value = current cell value × polygon value.
 - **Divide:** If polygon value ≠ 0, final cell value = current cell value ÷ polygon value. If polygon value = 0, the cell value remains unchanged.
 - **Minimum:** Final cell value = minimum of the current cell value and polygon value.
 - **Maximum:** Final cell value = maximum of the current cell value and polygon value.
 - **Average:** Final cell value = average of the current cell value and polygon value.

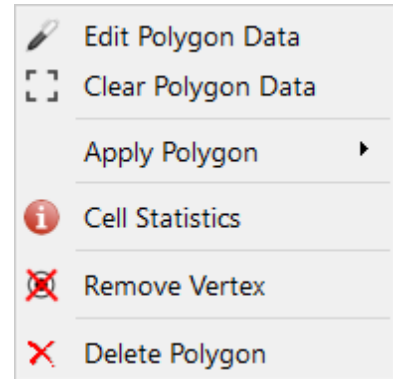


Figure 25. The Polygon Data Menu

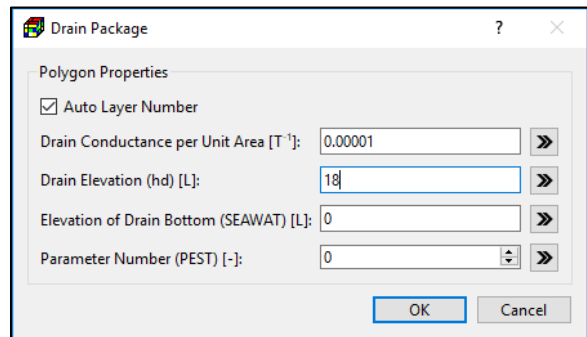


Figure 26. The Drain Package dialog box

¹⁹ In this case, the polygon's priority number and hole settings (see Figure 24) are ignored.

- **Cell Statistics:** Select this option to display statistics of the cells within the polygon, including:
 - Number of cells
 - Sum of cell values
 - Average cell value
- **Remove Vertex:** Select this option to remove a large white vertex along with its neighboring small vertices.
- **Delete Polygon:** Select this option to delete the polygon.

3.4.2.4. Pilot Points Input Method

The Pilot Point input method is available only when editing aquifer parameters such as HK, VK, Ss, Sy, etc.

The Pilot Points method is used to interpolate pointwise data to model cells using Kriging. Additionally, this method is frequently employed in nonlinear model calibration, which typically involves the following iterative steps:

1. Estimating hydraulic properties at selected (pilot) points.
2. Interpolating (and/or extrapolating) these pointwise hydraulic properties to model cells.
3. Running the groundwater model.
4. Comparing the model results with observed data and adjusting hydraulic properties at the pilot points accordingly.
5. Repeating the process until the model results closely match observed measurements.

Parameter estimation tools, such as PEST, are often used to automate this iterative calibration process. Kriging is commonly applied as the interpolation method. When using PEST in conjunction with pilot points, multiple Kriging processes can be conducted simultaneously, each with its own set of input and output files for a specific hydraulic property.

PM implements the Kriging process through two optional workflows. Both options are implemented on a layer-by-layer basis.

1. **ppk2fac1 and fac2real** (Doherty, 2020) – This is the default option.
 - The **ppk2fac1** program calculates Kriging factors from a set of pilot points to model cells in a specific layer.
 - The **fac2real** program then uses these Kriging factors to interpolate parameter values from the pilot points to the corresponding model cells in that layer.
2. **PLPROC** (Doherty, 2023)
 - The function `calc_kriging_factors_2d()` calculates Kriging factors.
 - The function `krige_using_file()` interpolates parameter values using the computed Kriging factors to assign values to model cells within the layer.

You can choose between these two options using the Kriging Tool dropdown list in the Preferences dialog box.

Both interpolation methods rely on user-defined variograms (see Section 3.4.2.4.5). Each zone (subarea) can have its own variogram, and interpolation does not occur across zone boundaries. A zone is defined as a group of model cells sharing the same negative Parameter/Zone Number. Within each zone, Kriging interpolation utilizes only the pilot points and Kriging parameters (variogram and associated settings) specified for that zone. If a zone contains only one pilot point, the zone is considered homogeneous, and no interpolation is performed. No interpolation is conducted in cells assigned a Parameter/Zone Number greater than or equal to 0.

The following sections describe how to generate pilot points, assign necessary input parameters, and apply the Kriging interpolation method (using either ppk2fac1 and fac2real or PLPROC). Additionally, settings for integrating the Pilot Points method with PEST are explained in detail.

3.4.2.4.1. Add Pilot Points

To activate the Pilot Points Input Method, first select a hydraulic property from the Parameters menu, and then select *Tools > Pilot Point Tool* or click the corresponding button on the toolbar. If the Pan Tool (Section 3.6.1) is not already activated, enable it by clicking its button on the toolbar or by selecting *Tools > Pan Tool*.

Once the Pilot Points Input Method is activated, follow the instructions below to add or delete pilot points, assign parameters to pilot points, and apply the Kriging method.

Adding Pilot Points

You can add pilot points to a model layer using one of three methods:

1. Add Individual Pilot Points (Ctrl+Click)

- Navigate to the desired model layer.
- Hold the Ctrl key and click at the desired location on the map to add a pilot point. Each pilot point appears on the map as a labeled circle.

2. Add a Group of Pilot Points (Ctrl+Drag)

- Navigate to the desired model layer.
- Hold the Ctrl key, click and drag the mouse to draw a window covering the target area, then release the mouse button to open the Add Pilot Points dialog box (Figure 27).
- Enter the pilot point parameters in the dialog box and click OK to add pilot points. The dialog box includes the following settings:
 - **AOI (Area of Interest):** Specifies the area in which pilot points will be placed. Options include:
 - Window: Represents the rectangular area drawn by dragging the mouse.

- Views: Any view (Section 2.1) created under Views in the TOC can define an irregular polygonal area of interest.
- **Row Interval and Column Interval:** Pilot points are placed within the selected area of interest in cells spaced by the specified row and column intervals.
- **PARVAL1, SCALE, and OFFSET:**
 - PARVAL1: Parameter value of the pilot point.
 - The value $[\text{PARVAL1} \times \text{SCALE} + \text{OFFSET}]$ is used by the model during Kriging interpolation. When running PEST, PARVAL1 represents the initial value of an adjustable parameter (see Section 3.9.5.2.1 for more detail).
- **PARLBNB, PARUBND, PARTRANS, PARCHGLIM, and PARGP:** These parameters are associated with PEST calibration settings (see Section 3.9.5.2.1 for further details).

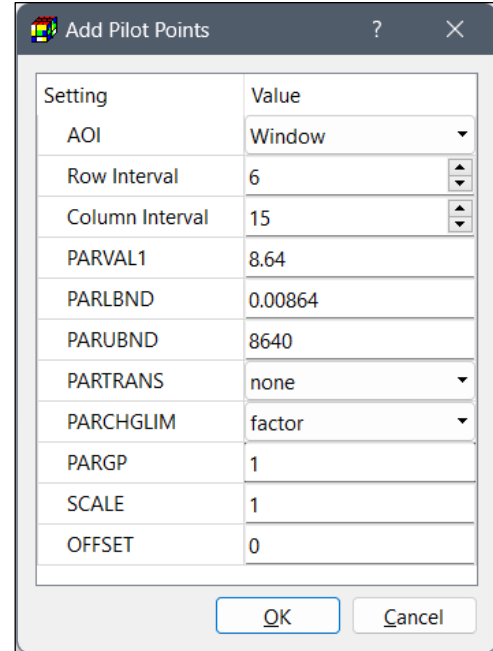


Figure 27. The Add Pilot Points dialog box

3. Import from Pilot Points Files

- Navigate to the desired model layer.
- Right-click anywhere on the Map Viewer to display the Pilot Point menu (Figure 28).
- Select Import & Append Pilot Points from the menu
- In the Open dialog box, select the desired pilot points files and click Open.
- Refer to Section 5.1.10 for the Pilot Points file format specification.

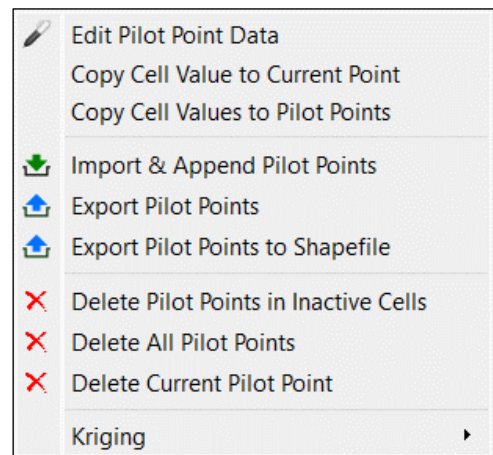


Figure 28. The Pilot Point menu

3.4.2.4.2. Export Pilot Points

Follow the steps below to export pilot points from a model layer:

1. Navigate to the model layer containing the pilot points you wish to export.
2. Right-click anywhere on the Map Viewer to open the Pilot Point menu (Figure 28).
3. Select Export Pilot Points from the menu.
4. In the displayed Save As dialog box, specify a filename and location for the exported Pilot Point file, and then click Save.

Tip: If many pilot points require modification, it may be more efficient to export the pilot points, modify the exported file externally, and then re-import the modified file. For details regarding the Pilot Points file format specification, refer to Section 5.1.10.

3.4.2.4.3. Delete Pilot Points

Pilot points can be deleted using one of the following methods:

1. Delete Individual Pilot Points (Shift + Click)
 - Navigate to the layer containing the pilot point(s) you want to delete.
 - Hold the Shift key and click directly on the pilot point to remove it.
2. Delete a Group of Pilot Points (Shift + Drag)
 - Navigate to the layer containing the pilot points you want to delete.
 - Hold the Shift key, click and drag to draw a window encompassing the desired pilot points, and release the mouse button. All pilot points within the selected window will be deleted.
3. Delete All Pilot Points in a Layer
 - Navigate to the layer from which all pilot points should be deleted.
 - Right-click anywhere on the Map Viewer to open the Pilot Point menu (Figure 28).
 - Select Delete All Pilot Points from the menu.
4. Delete Pilot Points in Inactive Cells
 - Navigate to the relevant model layer.
 - Right-click anywhere on the Map Viewer to open the Pilot Point menu (Figure 28).
 - Select Delete Pilot Points in Inactive Cells from the menu. Pilot points located within inactive cells will be deleted.
5. Delete Current Pilot Point
 - Right-click directly on the pilot point you want to delete to open the Pilot Point menu (Figure 28).
 - Select Delete Current Pilot Point from the menu to delete that specific pilot point.

3.4.2.4.4. Edit Pilot Point Data

Pilot point data can be edited individually or collectively using the Pilot Point Parameters dialog box (Figure 29). Follow the instructions below for different editing scenarios:

1. Edit an Individual Pilot Point

- Navigate to the model layer containing the pilot point you wish to edit.
- Right-click on the desired pilot point in the Map Viewer to display the Pilot Point menu (Figure 28).
- Select Edit Pilot Point Data to open the Pilot Point Parameters dialog box (Figure 29).
- Ensure that Apply To is set to Current Point, modify parameters as required, and click OK.

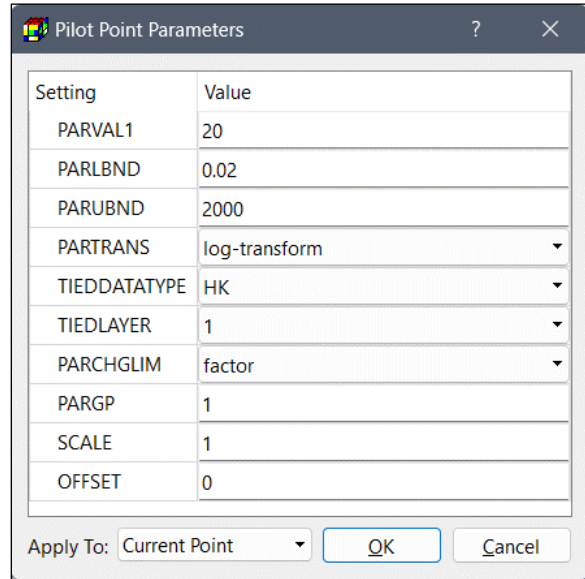


Figure 29. The Pilot Point Parameters dialog box

2. Edit a Group of Pilot Points

- Navigate to the model layer containing the pilot points you want to edit.
- Right-click anywhere on the Map Viewer to open the Pilot Point menu (Figure 28).
- Select Edit Pilot Point Data to open the Pilot Point Parameters dialog box (Figure 29).
- Ensure Apply To is set to Current Layer, a specific polygon, or a view:
 - If Current Layer is selected, parameter changes apply to all pilot points within the current layer.
 - If a polygon or a view is selected, parameter changes apply to all pilot points within the selected polygon or view.
- Adjust the parameters as required and click OK.

The parameters in the Pilot Point Parameters dialog box are used in conjunction with PEST calibration. Refer to Section 3.9.5.2 for the definition of the parameters. Pilot points can be tied to "parent" pilot points by setting the following variables appropriately:

- **PARTRANS:** Select *tied* to indicate that the pilot points will be tied to "parent" pilot points. Note that the parent pilot points themselves must not be fixed or tied.
- **TIEDDATATYPE:** Specifies the data type (e.g., HK or VK) of the parent pilot points. (PM does not verify whether the selected data type is actively used by the model.)
- **TIEDLAYER:** Select the model layer containing the parent pilot points.

For example, if you are editing VK, you can tie its pilot points to HK pilot points located in the first model layer by setting TIEDDATATYPE = HK and TIEDLAYER = 1. During parameter estimation with PEST, tied pilot points maintain the same ratio to their parent pilot points as defined by their initial values (i.e., PARVAL1).

3. Copy Cell Value to the Current Pilot Point

- Right-click directly on the pilot point to display the Pilot Point menu (Figure 28).
- Select *Copy Cell Value to Current Pilot Point*. This action copies the current cell's value into the pilot point's PARVAL1 field.
- If PARVAL1 is less than the parameter lower bound (PARLBND), PARLBND is automatically set equal to PARVAL1. Similarly, if PARVAL1 exceeds the parameter upper bound (PARUBND), PARUBND is automatically adjusted to equal PARVAL1.

4. Copy Cell Values to All Pilot Points in a Layer

- Navigate to the layer containing pilot points to which cell values will be copied.
- Right-click anywhere on the Map Viewer to open the Pilot Point menu (Figure 28).
- Select *Copy Cell Values to Pilot Points*. This action copies each cell's value into the PARVAL1 of the pilot point located within that cell.
- If PARVAL1 is less than the parameter lower bound (PARLBND), PARLBND is automatically set equal to PARVAL1. Similarly, if PARVAL1 exceeds the parameter upper bound (PARUBND), PARUBND is automatically adjusted to equal PARVAL1.

3.4.2.4.5. Edit Kriging Parameters

The Kriging Parameters dialog box (Figure 30) allows you to define variogram and interpolation parameters for zones within a model layer. These parameters are essential for using the ppk2fac1 & fac2real programs or the PLPROC utility, based on your selection in the Kriging Tool dropdown in the Preferences dialog box.

Integration with PEST is controlled via the Parameter Estimation section on the Options tab (Figure 31).

To open the Kriging Parameters dialog box, right-click in the Map Viewer and select *Kriging > Edit Parameters* from the Pilot Point menu (Figure 28). All parameters entered in this dialog box apply specifically to the current hydraulic property and current model layer. You can apply these parameters to all model layers by clicking Apply to All Layers in this dialog box.

1. Variogram Tab (Figure 30): Define variogram settings for each zone:

- **Active:** Activate this variogram for interpolation. Each zone requires an active variogram.
- **Zone:** Zone number²⁰ associated with this variogram. Interpolation occurs only within the zone.
- **Vartype:** Choose variogram type (Exponential, Spherical, Gaussian, or Power). The exponential type is recommended. See Part A of Groundwater Data Utilities (Doherty, 2020) for the definition of each of these variogram types.
- **A (Range):** Controls the spatial correlation range of the variogram.
- **Sill:** Variogram sill. Needed only if a non-zero nugget is specified.

²⁰ A model layer may be divided into zones by assigning *negative* parameter/zone numbers to model cells. Absolute values of those *negative* parameter/zone numbers are used as zone numbers for Kriging and Pilot Pilots.

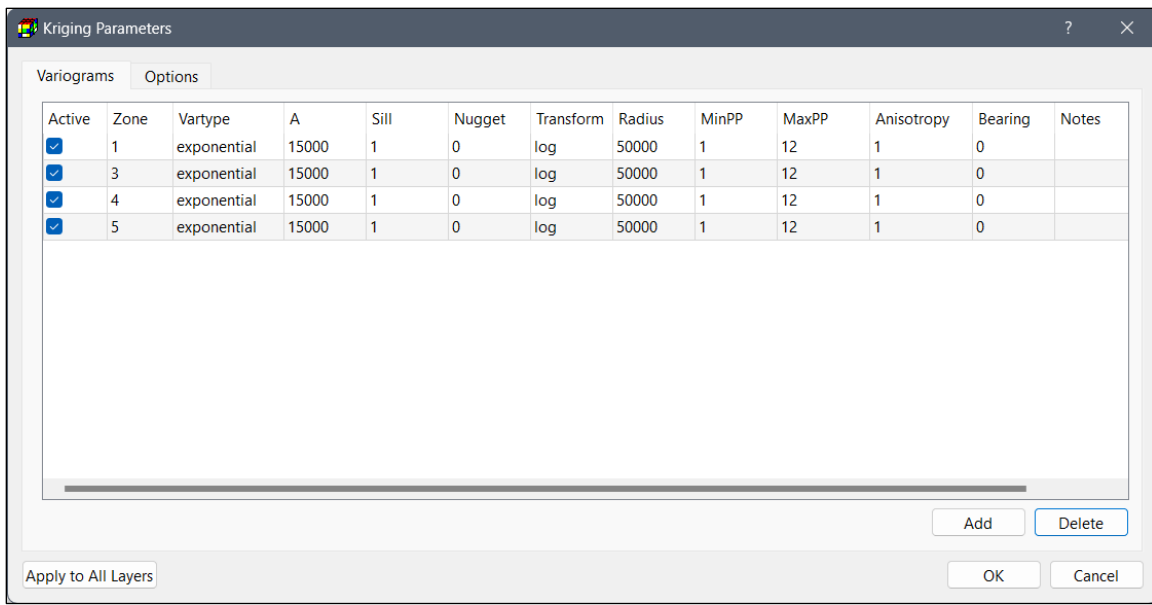


Figure 30. The Variogram tab of the Kriging Parameters dialog box

- **Nugget:** Specifies the nugget effect.
- **Transform:** Select *log* if parameters apply to log-transformed values (base 10); otherwise, select *none*.
- **Radius (Search Radius):** Maximum distance to search for pilot points during interpolation. A larger radius yields smoother interpolation, while a smaller radius allows local variation.
- **MinPP (Minimum Pilot Points):** Minimum number of pilot points required within the radius to interpolate a cell value. Behavior if unmet:
 - *ppk2fac1 & fac2real*: Uses the *Default Value* from the Options tab.
 - *PLPROC*: Stops interpolation and reports an error and then ceases execution
- **MaxPP:** Limits maximum pilot points used for interpolation, reducing computational time.
- **Anisotropy:** Ratio of variogram range in the *Bearing* direction to perpendicular direction (set to 1 for isotropy).
- **Bearing:** Direction (degrees from North) defining the axis of anisotropy.
- **Notes:** Field for user comments or notes.

2. Options Tab (Figure 31)

- **Kriging group**
 - **Upper Limit / Lower Limit:** Cell values exceeding these limits after interpolation are automatically adjusted to match the limits.
 - **Default Value (only for *ppk2fac1 & fac2real*):** Value assigned to cells where interpolation isn't performed (except for cells with parameter/zone numbers ≥ 0 , which remain unchanged).
 - **Required Distance (only for *ppk2fac1 & fac2real*):** If the distance between any pilot point pairs is less than the required distance, PPK2FAC1 will list the names of

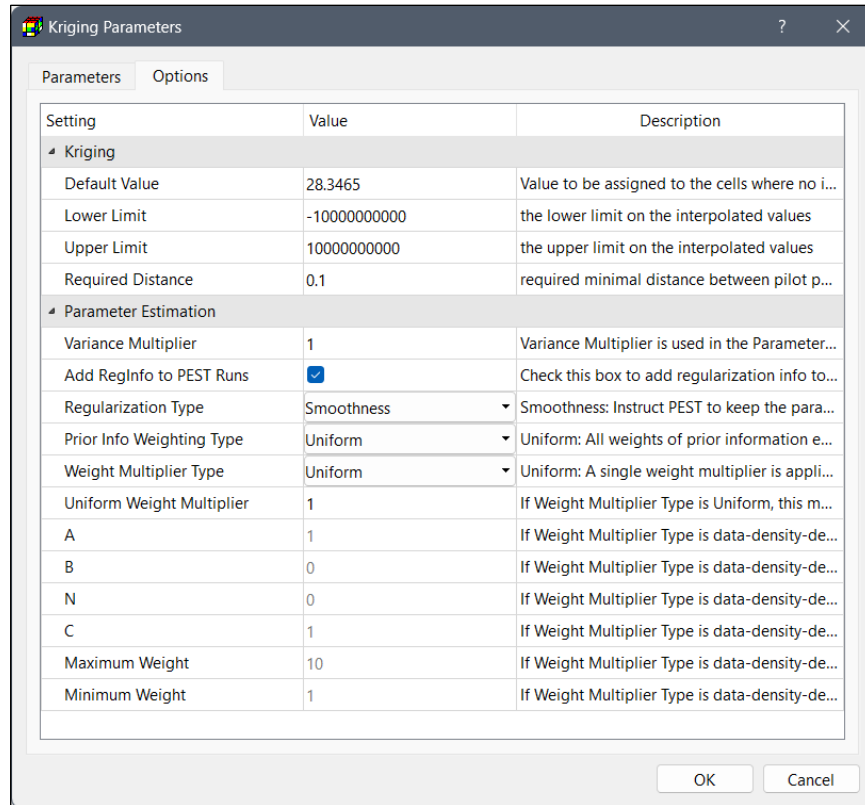


Figure 31. The Options tab of the Kriging Parameters dialog box

the points and then terminate execution. Set to zero to disable the distance check; a non-zero value prevents interpolation errors due to closely placed or duplicate pilot points.

- **Parameter Estimation group:** Settings in this section control integration with PEST and the handling of parameter estimation. Note: If the selected Kriging Tool **Error! Bookmark not defined.** is PLPROC, all options in this group except Variance Multiplier are ignored.
 - **Variance Multiplier:** Adjusts parameter variance values in the Parameter Uncertainty file.
 - **Add RegInfo to PEST Runs:** When checked, regularization prior information generated by PPKREG1 (Doherty, 2020) is added to the Pest Control file (created by PM). This helps PEST obtain a unique solution for inverse problems involving numerous pilot points. The following settings apply only if both *Use Pilot Point* and *Add RegInfo to PEST Runs* are enabled:
 - **Regularization Type:**
 - Smoothness: Instructs PEST to favor homogeneous parameter distributions.
 - Preferred **Value:** Instructs PEST to maintain original pilot point values where possible.
 - **Prior Info Weighting Type:**

- Uniform: Assigns equal initial weights (1.0) to all prior information equations, modifiable by a weight multiplier.
- Geostatistical: Calculates weights based on variogram properties as detailed in the PPKREG section of Groundwater Data Utilities (Doherty, 2020).
- **Weight Multiplier Type:**
 - Uniform: Applies a single multiplier to all prior information weights.
 - Data-density-dependent: Calculates weights based on proximity of pilot points to observation data points.
- **Uniform Weight Multiplier:** Applies when *Weight Multiplier Type* is set to Uniform, multiplying all prior information weights.
- **A, B, C, N, Maximum Weight, Minimum Weight:** Parameters used to calculate data-density-dependent prior information weights, as described in the PPKREG1 Section of Groundwater Data Utilities (Doherty, 2020). Minimum Weight and Maximum Weight define the allowable range for these calculated weights.


3.4.2.4.6. Apply Kriging

Once pilot points and all required parameters are entered, you can start the Kriging process by right-clicking anywhere on the Map Viewer and selecting *Kriging > Apply* from the Pilot Point menu (Figure 28).

The Kriging process involves the following automated steps, with progress displayed in the Kriging Output window:

1. All necessary input files for the selected Kriging tool (ppk2fac1 & fac2real or PLPROC) are generated and saved in the kriging subfolder within the model data folder.
2. A Windows batch file containing commands to execute the selected Kriging tool (ppk2fac1 & fac2real or PLPROC) is created and stored in the kriging subfolder.
3. The batch file is executed, producing output files.
4. Upon completion, the interpolated parameter values are automatically imported into the current model layer.

3.4.2.5. Managing Data of Multiple Stress Periods²¹

In MODFLOW, input data for each package is assumed constant throughout a given stress period but may change between periods. The Stress Period Data dialog box (Figure 32) enables effective management of data across multiple stress periods. Within the Data Editor, you can access this dialog box by clicking the  button. The features and functions of the dialog box are described below.

²¹ The data of certain packages, such as Horizontal Flow Barriers (HFB6), remain unchanged throughout the entire simulation. In such cases, the status bar will display "Time Independent", and all functions related to managing multiple stress periods are disabled. Other packages, such as Flow and Head Boundary (FHB) or Multi-Node Well 2 (MNW2), utilize time-series data instead of data defined individually for each stress period. For these packages, the Stress Period Data dialog box is deactivated.

1. **Stress Period Table:** The table summarizes the status of data for each stress period:

- **Data Column:**
 - **Checked:** Data for the stress period has been specified.
 - **Unchecked:** Data for the stress period has not been specified.
- **Use Column** (*editable only if data exists for that stress period*):
 - **Checked:** The specified data for the stress period will be used in the simulation.
 - **Unchecked:** The data from the previous stress period will be carried forward and used in the simulation.

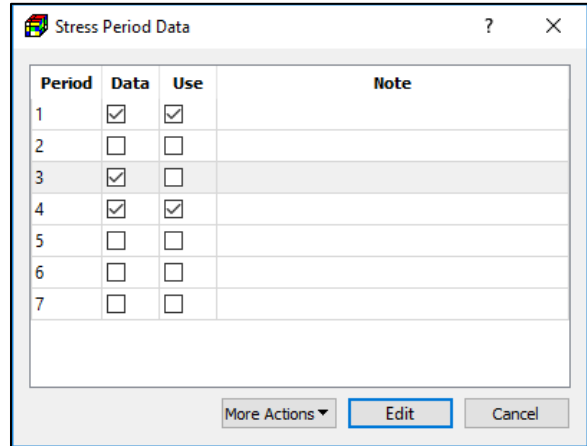


Figure 32. The Stress Period Data dialog box

- **Note Column:** Space provided to enter additional notes or comments.

2. **Edit button:** Click this button to edit data for the selected stress period.

3. **Cancel button:** Click this button to discard changes made to the *Use* flags and close the dialog box.

4. **More Actions:**

- **Copy:** Opens the Copy Data dialog box, allowing you to copy data from one stress period to another.
- **Interpolate:** Opens the Interpolate Data dialog box, enabling linear interpolation²² of cell-by-cell data between two specified stress periods. The interpolated values will change linearly over simulation time between the start and end periods. Interpolated values at the end of each interpolated stress period will be used in the model.

(Note: This option is available only for specific data types: Recharge (RCH), Evapotranspiration (EVT), Evapotranspiration Segments (ETS1), Unsaturated Zone Flow (UZF1), and concentration data associated with the Sink/Source Mixing (SSM) Package.)

- **Delete:** Deletes the specified data for the selected stress period.

Figure 32 illustrates an example scenario:

- Data for stress periods 1, 3, and 4 have been specified.
- Data specified for period 1 will be used for periods 1 through 3.
- Data specified for period 4 will be used from period 4 onward.


²² Linear interpolation applies to floating point values only. For integer values, such as parameter number, the values of the start period apply to all periods between the start and end periods.

- Data for period 3 is specified but will not be used in the simulation since its Use checkbox is unchecked.

3.5. File Menu

3.5.1. New

Select *New > Project in Cartesian CS* or *New > Project in Geographic CS* to start a new PM project in the Cartesian or Geographic coordinate system, respectively. If changes have been made to the current PM project, you will be prompted to save the changes to a PM file with the file extension “.pmproj”. The saved PM file will contain all model data and graphical objects except for the online basemaps.

The selected coordinate system becomes default and will be used when PM starts next time or when the  button on the toolbar is clicked. The selected coordinate system affects the features of PM as follows.

1. ***New > Project in Geographic CS:*** This initiates a new project using a geographic coordinate system. When this coordinate system is used, the following rules apply:
 - Online base maps may be used.
 - Models are placed on the world map by specifying the latitude/longitude coordinates of the model grid corners.
 - Observation points are also given in geographic coordinates.
 - Shapefiles, raster maps, ASCII rasters, CSV files that include spatial reference information files (i.e., files with the *prj* extension) will automatically be projected to the world map using the data included in those files.
 - Files exported from PM will include spatial reference information files when applicable. Those exported files may be imported to PM or most GIS applications.
2. ***New > Project in Cartesian CS:*** This starts a new project with a Cartesian (i.e., X, Y) coordinate system. When this coordinate system is used, the following rules apply:
 - Online base maps may not be used.
 - Models are placed on the Map Viewer by using the user-defined Cartesian coordinates of the model grid corners.
 - Coordinates of observation points, shapefiles, raster maps, ASCII rasters, CSV files are expected to be given in the same coordinate system as the model, i.e., coordinates will be taken as they are and will not be projected even if spatial reference information files (.prj) exist.
 - Files exported from PM will not include spatial reference information files.

3.5.2. Open

This menu item opens an existing PM file²³ in the Open Processing Modflow File dialog box. If changes have been made to the current session, you will be prompted to save the changes to a PM file with the file extension “.pmproj”. A PM file contains data of all loaded models and graphical objects, except for online base maps.

3.5.3. Recent Files

This menu allows you to open a PM file from the list of recently accessed files.

3.5.4. Save

Select this menu item to save the model data and settings to the currently opened PM file. For a new PM session, you will be asked to select a new PM file from the Save As dialog box. You should save a PM file in a folder with the full path of the folder containing only ASCII characters to avoid problems with model codes that may not be compatible with non-ASCII characters.

A backup of the original PM file is created before it is saved. The filename of a backup file consists of the original filename and the string “.~bak~YYYYMMDDHHMMSS”, where YYYY is year, MM is month, DD is day, HH is hour, MM is minute, and SS is second at which the backup file is created. The maximum number of backups is defined in the Preferences dialog box (Section 3.5.12). To restore a backup file, just rename it by removing the string “.~bak~YYYYMMDDHHMMSS”.

3.5.5. Save As

Select this menu item to display the Save As dialog box that allows you to save the model data and settings of all graphical objects to a new PM file. You should save a PM file in a folder with the full path of the folder containing only ASCII characters to avoid potential problems with model codes that may not be compatible with non-ASCII characters.

If you intend to create a new model variant, you should save the PM file in a new folder so that the generated model input and output files of the new variant do not overwrite the original model files.

3.5.6. Save As Reduced Size

Select this menu item to display the Save As dialog box that allows you to save the model data and settings of all graphical objects to a new PM file with a minimal amount of required disk space.

²³ A PM file may only be opened by a user at a time. As soon as a PM file is opened, a hidden “lock” file is created to prevent other users from opening the PM file. The lock file is in the same folder as the PM file. The filename of the lock file consists of the filename of the PM file and the string “.~lock~”. For example, the lock file for “MyModel.pmproj” is “MyModel.pmproj.~lock~”. The lock file will be deleted when the PM file or the application is closed. You may manually delete a lock file, in case it is not deleted correctly.

3.5.7. Create Model

This menu item opens the Create Model dialog box (Figure 33) that allows you to create a new model. When a model is created, PM makes the following changes:

1. A model item will be added to the Models group on the TOC. The newly created model will be set as the active model, and the model item will be highlighted in yellow. By default, the new model is configured to use MODFLOW-2005 and the Layer-Property Flow (LPF) Package. You can change the model settings by following the instructions outlined in Section 1.3.
2. A *model data folder* will be added to the folder where the PM file is located. The name of the model data folder is defined by Model ID that is shown in the Settings window of the model (Figure 2). All input and output files of the model are stored in that model data folder. See Section 5.3.1 for the names of those files.

The features of the Create Model dialog box are outlined below.

1. **Time Tab** defines the following temporal discretization parameters:
 - **Model Start Date/Time** is the start date/time of the flow simulation.
 - **Length of Simulation** is the length of the flow simulation.
 - **Time Unit** is the time unit of all model data value that involve time.
 - **Number of Stress Periods** is the number of stress periods in the flow simulation.

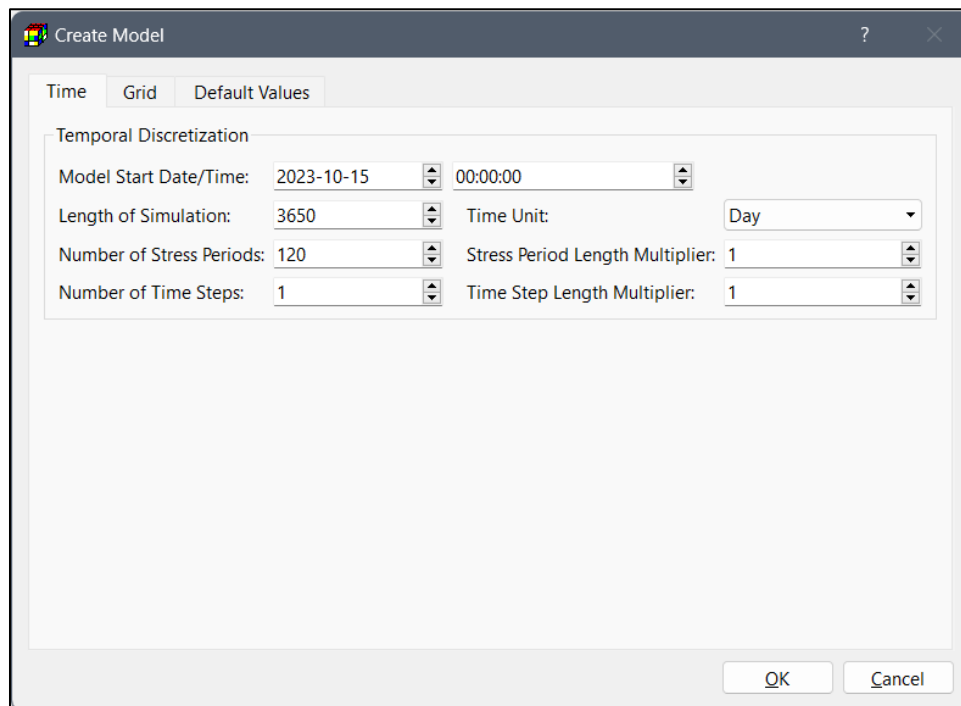


Figure 33 The Create Model dialog box

- **Stress Period Length Multiplier** is the ratio of the length of each stress period to that of the preceding stress period. The length of the first stress period, $\Delta T1$, is calculated as follows.

$$\Delta T1 = LS \cdot \left(\frac{SPMULT-1}{SPMULT^{NPER-1}} \right) \quad (1)$$

Where LS is the length of simulation, SPMULT is the stress period length multiplier, and NPER is the number of stress periods.

- **Number of Time Steps** is the number of time steps in each stress period.
- **Time Step Length Multiplier** is the ratio of the length of each time step to that of the preceding time step. For each stress period, the length of the first time step, $\Delta t1$, is related to PERLEN, NSTP, and TSMULT by the relation:

$$\Delta t1 = PERLEN \cdot \left(\frac{TSMULT-1}{TSMULT^{NSTP-1}} \right) \quad (2)$$

Where PERLEN is the length of a stress period, TSMULT is the time step length multiplier, and NSTP is the number of time steps of the stress period.


2. **Grid Tab** defines the following parameters for spatial discretization and georeferenced:

- **Spatial Discretization Group** contains the parameters that defines the extent of the model grid.
 - **Length Unit** is the length unit of all model data values that involve length.
 - **Top Elevation of the Model Grid** will be used as the top elevation of the first model layer.
 - **Number of Rows** is the number of model rows.
 - **Number of Columns** is the number of model columns.
 - **Number of Layers** is the number of model layers.
 - **Total Width of Rows** and **Total Width of Columns** defines the horizontal model extent. The width of individual rows is obtained by dividing the total width of rows by the number of rows. The width of individual columns is obtained by dividing the total width of columns by the number of columns.
 - **Total Thickness of Layers** defines the thickness of the model. The thickness of individual layers is obtained by dividing the total thickness of layers by the number of layers. The layer top elevation and bottom elevation are calculated based on the top elevation of the model grid and the thickness of individual layers. The elevation values may be modified with *Discretization > Layer Top Elevation* and *Discretization > Layer Bottom Elevation*.
- **Georeference Group**⁵ contains the parameters that are used to associate the model grid with a spatial location.
 - **Georeferencing Method** defines how the model grid is associated with a spatial location. Remember that the user-specified widths of model rows/columns are used in a simulation, even if the display of the model grid is stretched.

- **Four Grid Corners:** If this method is selected, the user-specified coordinates of the four model grid corners are used to place the model grid on the map.
 - **Lower-Left Grid Corner and Rotation:** If this method is selected, the grid azimuth and the coordinates of the lower-left grid corner are used to calculate the coordinates of the remaining grid corners. The result is then used to place the model grid on the map. The **Grid Azimuth** is the angle in degrees measured clockwise from a meridian to the axis along the model's row direction.
 - **Fit to Current View:** Click this button to switch the Georeferencing Method to “Latitude/Longitude Coordinates of Grid Corners” and set the coordinates of the grid corners to fit the current map viewer window.
3. **Default Values Tab** contains the default values for aquifer parameters. If a required aquifer parameter is not specified, its default value is used in the simulation.
 4. **OK:** Click OK to create the model. If the model is created successfully, a model item will be added to the Models group on the TOC and the model item will be highlighted in yellow indicating that the new model is set as the active model.

3.5.8. Import Model

This menu item opens the Import Model dialog box (Figure 34) that allows you to import an existing MODFLOW model that are stored in the standard format as described in the user guides of MODFLOW-96 (Harbaugh, et al., 1996), MODFLOW-2000 (Harbaugh, et al., 2000) or MODFLOW-2005 (Harbaugh, 2005). The features of the Import Model dialog box are discussed below.

1. **Model Name File:** Click on the  button and then select a name file from the Open Name File dialog box. After a model is created, a model item will be added to the Models group on the TOC. The filename of the name file will be used as the name of the model item.
 - PM imports a MODFLOW or a SEAWAT model through a Name File, which specifies the names of the input and output files, associates each file name with a unit number and identifies the packages that will be used in the model. Most packages (except OC, SFR2²⁴, and LMT²⁵) of MODFLOW listed in Section 5.3.1 and all packages of MT3D and SEAWAT listed in Sections 5.3.2.1 and 5.3.4.1 may be imported.
 - To import a MT3D model, you must import the corresponding MODFLOW flow model first, and then subsequently import the MT3D transport model by selecting *Models > MT3D/SEAWAT > Settings*. See Section 3.9.3.1 for details.
2. **Time Tab** defines the following temporal discretization parameters:
 - **Model Start Date/Time** is the start date/time of the flow simulation.

²⁴ Supported SFR2 package files must have the REACHINPUT option activated, NSFRPAR = 0 and NPARSEG = 0. See <https://water.usgs.gov/ogw/modflow/MODFLOW-2005-Guide/index.html?sfr.htm> for the meaning of those parameters.

²⁵ Import of an LMT package file is not necessary.

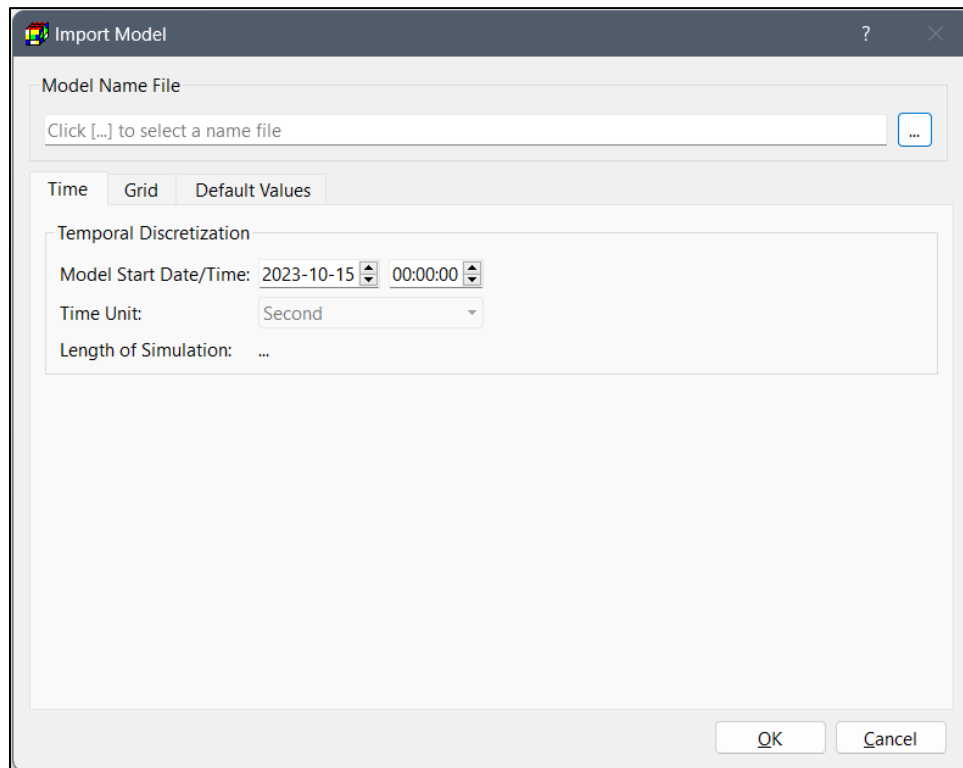



Figure 34 The Import Model dialog box

- **Time Unit** is the time unit of all model data value that involve time. The Time Unit is obtained from the imported model.
 - **Length of Simulation** is the length of the flow simulation of the imported model.
3. **Grid Tab** defines the following parameters for spatial discretization and georeferenced:
- **Spatial Discretization Group** contains the parameters that defines the extent of the model grid.
 - **Length Unit** is the length unit of all model data values that involve length.
 - **Number of Rows** is the number of model rows.
 - **Number of Columns** is the number of model columns.
 - **Number of Layers** is the number of model layers.
 - **Total Width of Rows** and **Total Width of Columns** defines the horizontal extent of the imported model.
 - **Georeference Group**⁵ contains the parameters that are used to associate the model grid with a spatial location.
 - **Georeferencing Method** defines how the model grid is associated with a spatial location.
 - **Latitude/Longitude Coordinates of Grid Corners:** If this method is selected, the user-specified coordinates of the four model grid corners are used to place the model grid on the map.

- **Grid Azimuth and Coordinates of the Lower-Left Grid Corner:** If this method is selected, the grid azimuth and the coordinates of the lower-left grid corner are used to calculate the coordinates of the remaining grid corners. The result is then used to place the model grid on the map. The **Grid Azimuth** is the angle in degrees measured clockwise from a meridian to the axis along the model's row direction.
4. **Default Values Tab** contains the default values for aquifer parameters. If a required aquifer parameter is not specified, its default value is used in the simulation.
 5. **OK:** Click OK to start the import process. The following actions will be taken during the import process.
 - A model item will be added to the Models group on the TOC. The base filename of the Name file will be used as the name of the model item.
 - A "Model ID" will be assigned to the model item. A subfolder, using the Model ID as its name, will be created in the folder where the PM file is located. All future model input and output files, except the PM file, will be stored in this subfolder.
 - PM will import the input files listed in the Name File. If an input file cannot be found in its given path, PM will search for the input file in the folder where the selected Name File is located. Detailed information about the import process will be stored in the log file "modflow_import_log.txt".
 - If the model import is completed successfully, the model item will be highlighted in yellow indicating that the newly imported model is set as the active model. By clicking on the model item, you have access to the model's Settings windows. See previous section for details.

3.5.9. Save Image

Displays the Save Image dialog box (Figure 35) that allows you to save the map as an image file. Click on the  button to set the image file name in the Save Image dialog box. Use the Magnification dropdown box to select a magnification factor of the current map. Set a larger value for a higher output resolution²⁶. The saved image files can be used as raster maps, see Section 2.4 for details.

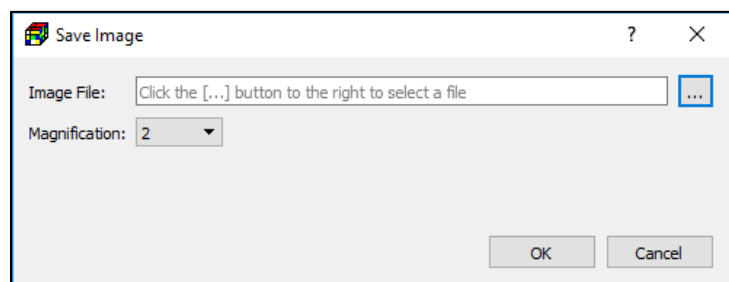


Figure 35 The Save Image dialog box

Two additional files with the common base filename and the following extensions are created:

1. .pgw, .jgw, or .tfw – contains the World file⁷ for the image saved in the PNG, JPEG, or TIFF format, respectively. If the geographic coordinate system is used (see Section 3.5.1 for details), the coordinates in the world file are expressed in the WGS 1984 World Mercator

²⁶ Setting a higher magnification factor does not change or improve the resolution of the base maps.

coordinate system. If the Cartesian coordinate system is used, the coordinates in the world file are expressed in the user-specified Cartesian coordinate system.

2. .prj – contains the spatial reference information for the WGS 1984 World Mercator coordinate system. This file is created only if the Geographic coordinate system is used.

3.5.10. Animation

Displays the Animation dialog box (Figure 36) that allows you to run animations with the model data and optionally save animations into video files or sequences of image files.

1. Time Control

- **Start:** Start date/time of an animation sequence.
- **End:** End date/time of an animation sequence.
- **Steps:** Number of animation steps between Start and End. For each step, the corresponding date/time is determined by linear interpolation

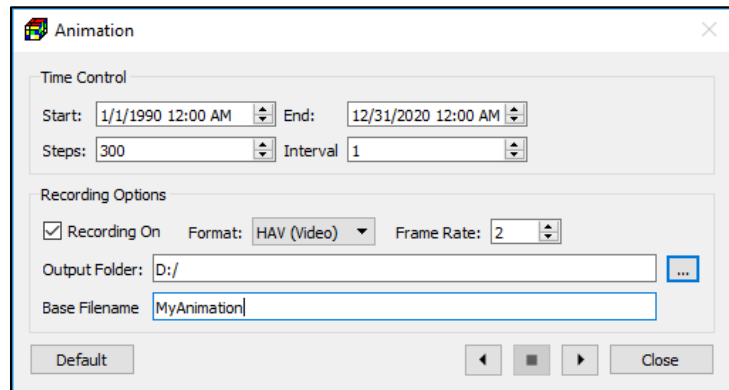


Figure 36 The Animation dialog box

between Start and End, and then the model results at the date/time are determined and used for visualization with the following rules:




- If a step lies before first model result date/time, the first model result will be used for visualization.
 - If a step lies after the last model result date/time, the last model result will be used for visualization.
 - If a step lies between date/time of two model results, those results will be linearly interpolated to the step. The interpolated values will be used for visualization.
- **Interval:** A video frame is produced when the current animation step is an even multiple of Interval.

2. Recording Options

- **Recording On:** Check this box to turn on recording.
- **Format:** Output format. Select a video format (AVI or HAV) or image format (PNG or JPG). The recorded video frames or images will have the same resolution²⁷ as the map viewer. AVI is a Microsoft video format that can be played back on most video players. HAV (High quality Audio Video) is a lossless compressed format that can be played back with the free Imagen multimedia player and can be converted to other formats

²⁷ It is known that Windows Media Player on Windows 7 or earlier may encounter playback problems with videos that have more than 1080 horizontal lines. It is recommended to resize the Map Viewer to a resolution less than 1080p (i.e., 1920 x 1080) before recording videos. The size of the Map Viewer is displayed on the status bar when resizing the Main Window of PM.

with the VideoMach software. Both software packages are available on www.gromada.com.

- **Frame Rate:** number of frames per second when a recorded video file is played back. The total duration (in seconds) of the recorded video file is approximately $Steps/Interval/Frame\ Rate$.
 - **Output Folder:** click the  button to select a folder where the video or image files will be saved.
 - **Base Filename:** This the base filename (i.e., without a file extension) of output files. The appropriate file extension will be determined based on the selected format, and appended to the base filename.
3. : Click this button to play the animation sequence forwards from *Start* to *End*. Output file(s) will be created or updated if *Recording On* is checked. Make sure that the existing output file(s) are not opened in other applications that may prevent those file(s) from being updated.
 4. : Click this button to play the animation sequence backwards from *End* to *Start*. Output file(s) will be created or updated if *Recording On* is checked. Make sure that the existing output file(s) are not opened in other applications that may prevent those file(s) from being updated.
 5. Default: Click this button to set default values to all controls, i.e., *Start* to the model start date/time, *End* to the model end date/time, *Steps* to 10, *Recording* off, *Format* to HAV, *Frame Rate* to 4, etc.

3.5.11. Print

Select this menu item to open the Print dialog box. Its right-hand panel provides a print preview. You can adjust the print settings with the controls on the left-hand panel. Optionally, you can enter a caption and load a logo (saved in the .png or .jpg formats). Click on the Print button on the Top-Left corner of the dialog box to print.

3.5.12. Preferences

Displays the Preferences dialog box (Figure 37) with four tabs that define various settings. The settings marked with the asterisk (*) symbol are stored with the current PM file. All other settings apply to the current computer.

1. **Path Tab:** This tab displays a table that specifies the file paths to the executable files of the supported models and applications. PM installs these executables in their default directories. When a model is run, PM launches the associated executable—either directly or via a batch file containing the necessary run commands. To update the path for a specific model or application, double-click the corresponding entry in the table and select a new executable from the Select Model Executable File dialog box. Note: Some executables have been recompiled after modifications were made to their original source code. For more information, refer to the readme.txt files in their respective folders.

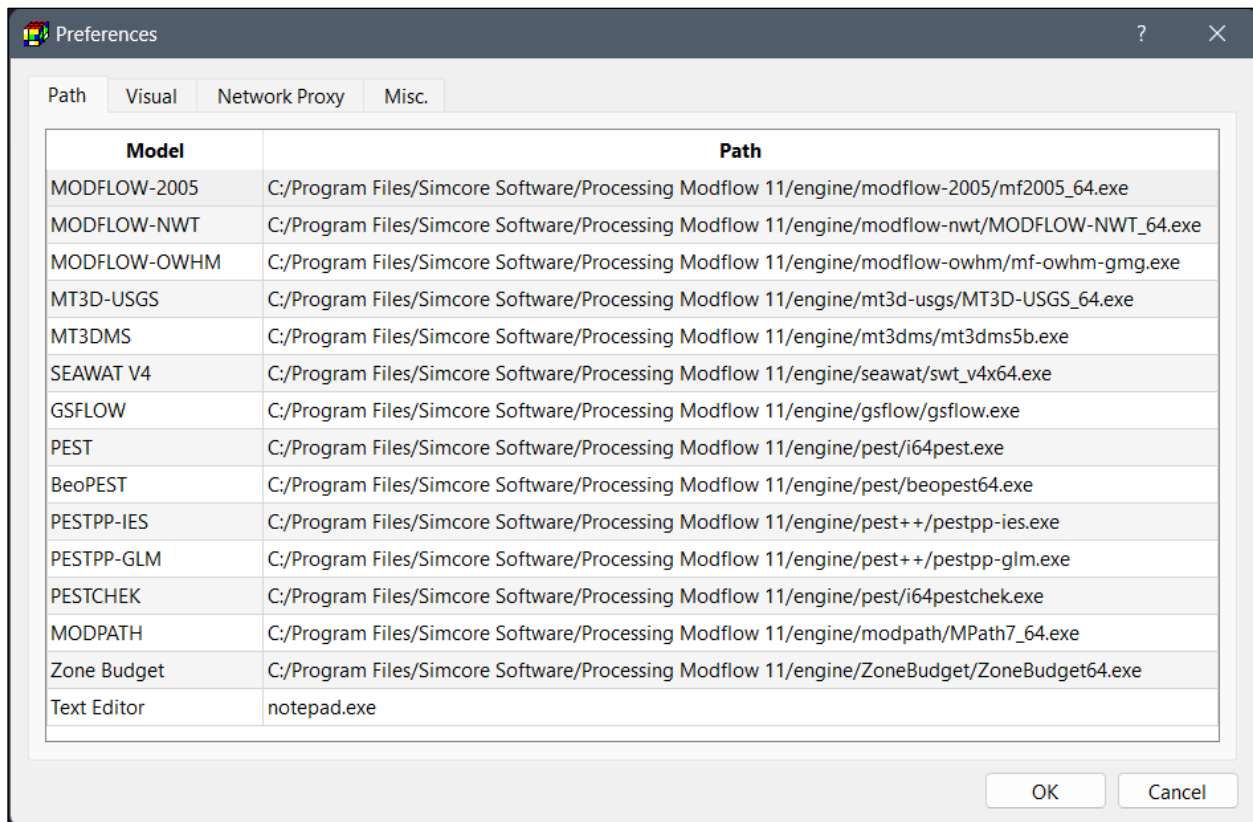


Figure 37. The Preferences dialog box

- MODFLOW-2005:** MODFLOW-2005 is a core version of the USGS's three-dimensional (3D) finite-difference groundwater model. For details, visit <https://www.usgs.gov/software/modflow-2005-usgs-three-dimensional-finite-difference-ground-water-model>
- MODFLOW-NWT:** MODFLOW-NWT is a Newton-Raphson formulation for MODFLOW-2005 to improve solution of problems involving drying and rewetting nonlinearities of the unconfined groundwater-flow equation. For details, visit <https://water.usgs.gov/ogw/modflow-nwt/>
- MODFLOW-OWHM:** The MODFLOW One-Water Hydrologic Flow Model is a MODFLOW-2005 based integrated hydrologic model designed for the analysis of conjunctive-use management. The term “integrated” refers to the tight coupling of groundwater flow, surface-water flow, landscape processes, aquifer compaction and subsidence, reservoir operations, and conduit (karst) flow. This fusion results in a simulation software capable of addressing water-use and sustainability problems, including conjunctive-use, water-management, water-food-security, and climate-crop-water scenarios. MODFLOW-OWHM is backward-compatible with MODFLOW-2005 and MODFLOW-NWT. For details, see <https://www.usgs.gov/software/modflow-one-water-hydrologic-flow-model-conjunctive-use-simulation-software-mf-owhm>

- **MT3DMS:** MT3DMS is a modular three-dimensional multispecies transport model for simulation of advection, dispersion and reactions of solutes in groundwater systems. For details, visit

<https://hydro.geo.ua.edu/mt3d/index.htm>
- **MT3D-USGS:** MT3D-USGS is a USGS updated release of the groundwater solute transport code based on MT3DMS. MT3D-USGS includes new transport modeling capabilities to accommodate flow terms calculated by MODFLOW packages that were previously unsupported by MT3DMS and to provide greater flexibility in the simulation of solute transport and reactive solute transport. For details, visit

<https://www.usgs.gov/software/mt3d-usgs-groundwater-solute-transport-simulator-modflow>
- **SEAWAT V4:** SEAWAT is a generic MODFLOW/MT3DMS-based computer program designed to simulate three-dimensional variable-density groundwater flow coupled with multi-species solute and heat transport. The program has been used for a wide variety of groundwater studies including those focused on brine migration in continental aquifers as well as those focused on saltwater intrusion in coastal aquifers. For details, visit

<https://www.usgs.gov/software/seawat-computer-program-simulation-three-dimensional-variable-density-ground-water-flow>
- **GSFLOW:** GSFLOW is a coupled surface water and groundwater flow model based on the integration of PRMS and MODFLOW-NWT, respectively. GSFLOW was developed to simulate coupled surface-water/groundwater flow by simultaneously simulating flow across the land surface, within streams and lakes, and within subsurface saturated and unsaturated materials. For details, visit

<https://www.usgs.gov/software/gsfow-coupled-groundwater-and-surface-water-flow-model>
- **PEST:** This is the standard version of PEST that runs in a single thread. PEST is the industry standard software package for parameter estimation and uncertainty analysis of complex environmental and other computer models. While a parallel version of PEST exists, it is not currently supported by PM. For details, visit

<https://www.pesthomepage.org/>
- **BeoPEST:** BeoPEST is a special version of Parallel PEST inspired by Beowulf Clusters. A Beowulf Cluster contains several computers that are tied together using a reasonably fast network. When running BeoPEST within PM on a multicore computer, it is configured to run solution agents on individual CPU cores to accelerate the parameter estimation process (by solving the Jacobian matrix parallelly). For details, visit


<https://pesthomepage.org/programs>

- **PESTPP-IES:** PESTPP-IES is a part of the PEST++ (Welter, et al., 2015; White, 2018; White, et al., 2020), that is a software suite aimed at supporting complex numerical models in the decision-support context. The main purpose of PESTPP-IES is to produce an ensemble of calibrated parameter sets. PESTPP-IES uses a built-in parallel run manager. For details, see <https://github.com/usgs/pestpp>
- **PESTPP-GLM:** PESTPP-GLM is a part of the PEST++ (Welter, et al., 2015; White, 2018; White, et al., 2020), that is a software suite aimed at supporting complex numerical models in the decision-support context. PESTPP-GLM is the original member of the PEST++ suite; it was originally named “PEST++” itself. Like PEST, PESTPP-GLM undertakes non-intrusive, highly parameterized inversion of an environmental model. As such, it can be considered as a direct replacement for PEST. PESTPP-GLM uses a built-in parallel run manager. For details, see <https://github.com/usgs/pestpp>
- **PESTCHEK:** PESTCHEK is a utility program that checks an entire PEST input dataset for correctness and consistency. For details and for additional utilities, visit https://www.pesthomepage.org/PEST_Uutilities.php
- **MODPATH:** MODPATH is a particle-tracking post-processing program designed to work with MODFLOW. The included MODPATH version 7 is the fourth major release since its original publication. Note that MODPATH works with MODFLOW-NWT²⁸ provided that the option flag IPHDRY in *Models > Modflow > Output Control* is checked. For details about MODPATH, visit <https://www.usgs.gov/software/modpath-particle-tracking-model-modflow>
- **Zone Budget:** ZONEBUDGET is a program for computing subregional water budgets for MODFLOW. For details, visit <https://www.usgs.gov/software/zonebudget-program-computing-subregional-water-budgets-modflow-groundwater-flow-models>
- **Text Editor:** The text editor is used to display simulation log and certain result files that are saved in the text format. Most text editors, such as NotePad++ or EmEditor, may be used instead of the Windows’s default Notepad application.

2. Visual Tab

- **Map Viewer**
 - **Display Coordinates As:** This dropdown box defines the types of coordinates to be displayed on the status bar.

²⁸ For details about the compatibility of MODPATH with MODFLOW-NWT, see this page <https://water.usgs.gov/water-resources/software/MODPATH/UsingModpathWithModflow-NWT.txt>

- **Unit of Ground Surface Elevation:** This defines the unit of the ground surface elevation at the mouse cursor's location on the map. Elevation values are provided for reference purposes only and do not represent precisely measured or surveyed values.
- **Map Scale Color:** This defines the color of the map scale.
- **Map Viewer Background Color:** This defines the background color of the Map Viewer.
- **Raster Map Image Size:** Size limit of a raster map (Section 2.4) image in mega pixels. If an image exceeds this size, it is scaled down.
- **Display Map Scale:** Check this box to display the scale on the lower-left corner of the Map Viewer; clear this box to hide the scale.
- **Set Current View as Default:** Click this button to set the current view extent as the default view extent when PM is started. This button is deactivated when the  button on the toolbar is depressed.
- **On-Map Popup Labels**
 - When the mouse hovers over some visual objects, such as flow vectors, their detailed information can be displayed with a pop-up label. The colors and font size of the pop-up label are defined in this group.
- **Model**
 - **Grid Cursor Color and Opacity:** These controls define the color and opacity of the grid cursor. For details about the grid cursor, see Section 3.4.3.4.2
 - **Polyline Color, Opacity and Line Width:** These controls define the color, opacity, and line width of polylines created by the Polyline input method (Section 3.4.2.2). The value of opacity ranges from 0 to 100, with 0 being transparent and 100 being opaque.
 - **Polygon Color, Opacity and Line Width:** These controls define the color, opacity, and line width of polygons created by the Polygon input method (Section 3.4.2.3).
 - **Highlight Opacity and Auto Clear Highlight:** Within the data editor (Section 3.4.2), you can double-click on a cell to highlight all other cells that have the same value as the clicked cell. The value of Highlight Opacity controls the opacity of the highlight mask. The color of the highlight mask changes with repeated double-clicks. When the **Auto Clear Highlight** box is checked, the highlight mask will be cleared automatically whenever the model grid is drawn or redrawn.
 - **Auto Adjust Layer Elevation:** When the top of bottom elevation of a model layer is modified, the elevation of the neighboring layers will be automatically adjusted to match when the **Auto Adjust** box is checked.
 - **Display Pilot Points in All Input Methods:** When this box is checked, pilot points are always displayed in the Data Editor regardless of the selected input method.
 - **Display Polylines in All Input Methods:** When this box is checked, polylines are always displayed in the Data Editor regardless of the selected input method.
 - **Display Polygons in All Input Methods:** When this box is checked, polygons are always displayed in the Data Editor regardless of the selected input method.

- **Value Display Precision:** This is the maximum number of significant digits that is used to display model values on the Properties window.
- **Dry Cell Tolerance:** If the simulated saturated thickness of a cell (in an unconfined or a convertible layer) is less than this value, the cell is displayed as a dry cell and is excluded from the storage calculation done by *Models > Modflow > Volume Summary*. In addition, head contours and flow vectors are not drawn for dry cells. This setting is designed for graphical purposes, use with caution. Enter 0 (zero) to deactivate this setting. Enter a positive value to activate this setting.
- **Synchronize models:** If this box is checked and all displayed models have the same number of layers and temporal discretization, the toolbar's stress period, time step, and layer controls will apply to all displayed models. Otherwise, these controls will affect only the active model.

3. Network Proxy Tab

- **Proxy Type:**
 - **No Proxy:** Choose this if you don't want to use a proxy.
 - **Auto-Detect:** Choose this if you want PM to automatically detect the proxy settings.
 - **Manual:** Choose this if you want to configure the proxy settings manually. PM uses the SOCKS5 protocol to connect to the proxy server.
- **Proxy Settings:** If the selected Proxy Type is *Manual*, you need to provide the proxy server details as follows.
 - **Host:** address of the proxy server
 - **Port:** port of the proxy server
 - **User Name:** user name for proxy authentication
 - **Password:** Password for proxy authentication.

4. Misc. Tab


- **Before Running a Model:** Defines the action to be taken before a model run is commenced.
- **After Deleting a Model:** Defines the action to be taken after a model is deleted.
- **PEST Version:** Defines the PEST version to be used for the parameter estimation process. *PEST Version* is ignored if the PEST Operation Mode (Section 3.9.5.2) is set to *Stochastic (PESTPP-IES)*. In that case, PESTPP-IES is used.
- **Number of Agents:** This is the number of BeoPEST, PESTPP-IES, or PESTPP-GLM instances that are used during a parameter estimation process. The default value is half of the CPU threads.
- **Kriging Tool:** This defines the tool that is used to interpolate values from pilot points to model cells. *PLPROC* stands for "parameter list processor" and is a member of the PEST suite. *PPK2FAC1* & *FAC2REAL* are members of the PEST groundwater utility suite. PLPROC supports more pilot points. However, unlike PPK2FAC1, PLPROC does not generate regularization information files, which can be used by the PPKREG1 utility

add prior information equations that encompass preferred-value or preferred-difference regularization to a PEST control file.

- **Auto Cleanup:** Delete the files of the BeoPEST, PESTPP-IES, or PESTPP-GLM agents after completion of a run.
- **Port:** This is the TCP/IP port number that is used by BeoPEST, PESTPP-IES, or PESTPP-GLM to communicate between the master and agent instances. The port number can be any unused port on the host where the master is run. Port numbers are positive integers between 1024 and 65536. Typically, a port number such as 4004 is unused.
- **Output Width:** This is the output width (in pixels) of charts.
- **Font Size:** This sets the font size of the axis labels and tick labels of charts.
- **Maximum Number of Backups:** This is the maximum number of backups of the currently loaded PM file. A backup of the loaded PM file is created before it is saved.
- **Start Data Editor in the first stress period:** If this box is checked, the Data Editor will start in the first stress period or the first stress period with existing data, whichever is greater. If this box is not checked, the Data Editor will start in the current stress period.
- **Display options to edit or deactivate a package before entering Data Editor:** If this check box is checked, PM will display a prompt box before entering the data editor for editing a selected package. The prompt box allows you to deactivate or activate/edit the selected package. If this box is not checked, the selected option is assumed to be activate/edit.

3.6. Tools Menu

3.6.1. Pan Tool


To activate the Pan Tool, click on the Pan Tool icon  in the toolbar or select *Tools > Pan Tool*. The Pan tool has the following major functions.

1. Move the map view in any direction: Place the mouse cursor within the Map Viewer and drag to move the view. Alternatively, the w, e, d, c, x, z, a, and q keys can be used to move the view towards the north, northeast, east, southeast, south, southwest, west, and northwest, respectively.
2. Move the grid cursor:
 - Move the grid cursor to a cell by left clicking on that cell.
 - Drag the grid cursor with the mouse.
 - Use the arrow keys to move horizontally. Use the Shift + PageUp and Shift + PageDown keys to move to the upper or lower layer.
3. Move or rotate the model grid²⁹: Select *Discretization > Grid* to enter the Grid Editor, and then follow these steps:

²⁹ You can also define the position of a model grid with the Settings window of the model, see Figure 2.

- To move the model grid with shift+drag: Place the mouse cursor over the grid cursor, press and hold the Shift-key and the left mouse button, move the mouse to drag the model grid, and then release the Shift -key and left mouse button to place the model grid.
 - To rotate the model grid with ctrl+drag: Place the mouse cursor over the grid cursor, press and hold the Ctrl-key and left mouse button, move the mouse to rotate the model grid, and then release the Ctrl-key and left mouse button to place the model grid. The center of rotation is the lower-left corner of the unrotated grid.
4. Zoom in: If your mouse has a scroll wheel, it can be used to zoom in by scrolling it forward. Alternatively, place the mouse cursor within the Map Viewer, press and hold the right button, and move the mouse forward. Release the button when you reach the desired zoom level.
 5. Zoom out: If your mouse has a scroll wheel, it can be used to zoom out by scrolling it backward. Alternatively, place the mouse cursor within the Map Viewer, press and hold the right mouse button, and move the mouse backward. Release the button when the desired zoom level is reached.

3.6.2. Zoom Tool


To activate the Zoom Tool, click on the Zoom Tool icon  in the toolbar or select *Tools > Zoom Tool*. The Zoom tool has the following major functions.




1. Move the map view in any direction: Press the arrow keys to move the map view toward their respective directions. The w, e, d, c, x, z, a, and q keys can also be used to move the view towards north, northeast, east, southeast, south, southwest, west, and northwest, respectively.
2. Move the grid cursor to a desired cell by left clicking on that cell.
3. Move the model grid with shift+drag: Same as the Pan tool described above.
4. Rotate the model grid with ctrl+drag: Same as the Pan tool described above.
5. Zoom in: To zoom in, position the mouse cursor on the corner of the desired area, press and hold down the left button, drag the mouse cursor to the opposite corner of the desired area, and release the left button. If your mouse has a scroll wheel, it can be used to zoom in by scrolling it forward.
6. Zoom out: If your mouse has a scroll wheel, it can be used to zoom out by scrolling it backward.

3.6.3. Observation Tool

The Observation tool is used to define the observation points, and observation values pertaining to the observation points. The data can be used for model calibration³⁰ and displayed in the

³⁰ Head, Drawdown, and Streamflow observation may be used in conjunction with PEST for model calibration. Streamflow observation works with PEST only if an observation point is placed within a stream reach of the SFR2 Package, and that stream reach is designated as a gaging station, see Figure 92 for details.

Scatter and Line charts. To activate the Observation Tool, click on the Observation Tool icon  in the toolbar or select *Tools > Observation Tool*.

1. **Add observation points (ctrl+click):** To add observation points to the active model, move to the desired layer with  or  buttons on the toolbar and then ctrl+click on the desired locations on the Map Viewer. The color of the observation points matches with the color assigned to the layer (see Section 3.7.3 for the Layer Properties).
2. **Remove an observation point (shift+click or shift+drag):** Shift+click on the observation point in the Map Viewer or shift+drag a window encompassing the observation points that should be deleted, then release the mouse button. Alternatively, right click on the observation point in the Map Viewer to open the Observation dialog box (Figure 38), select the observation point in the table, and then click Delete in the dialog box to delete the selected points.
3. **Move observation points:** Use the mouse to drag observation points to desired locations. If observation points are not draggable, check the “draggable” box in the Settings window of the Observation item on the TOC.
4. **Edit observation data:** The Observation dialog box (Figure 38) can be used to enter and/or edit observation data. This dialog box can be opened by either right-clicking on an observation point in the Map Viewer or by right-clicking on the Observation Points group on the TOC, and then selecting Edit Data. The features of the dialog box are as follows.
 - **Observation Point (left pane of the dialog box)**
 - **Table:** the table displays the observation points and their properties.
 - : Check/uncheck the boxes in this column to turn on/off observation points.

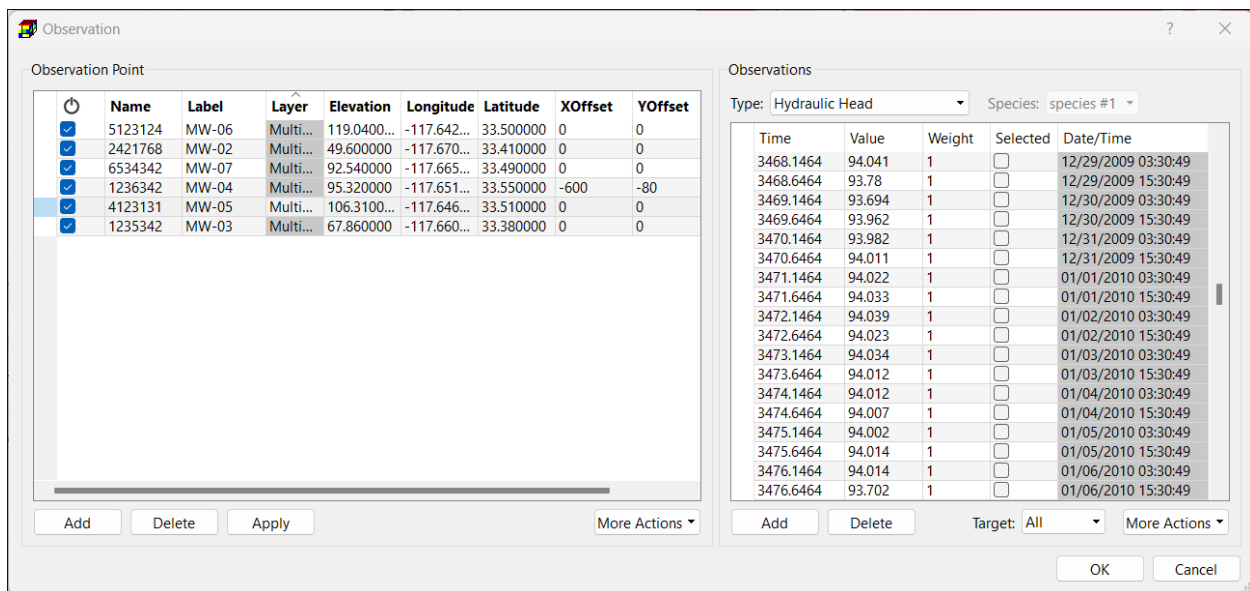


Figure 38. The Observation dialog box

- **Name:** This column displays the observation point names. The names must be unique. Each name must be of 16 characters or less in length (shorter is recommended). Space and the tilde (~) character are not allowed. When you click OK, PM will remove spaces and tildes from Name and will make sure that all names are unique by suggesting new names if needed.
- **Label:** This column contains the observation point labels that are displayed on the map. Use “\n” to move to next line; “\u” to move to previous line; and “\” for backslash. Only ASCII characters with values from 32 to 126 are supported.
- **Layer:** Double-click on the cell of the respective well to edit its layer weights in the Layer Weight dialog box (Figure 39). A single-layer observation point is defined by setting a positive Weight value to one and only one layer. A multi-layer observation point is defined by assigning positive Weight values to more than one layer.

PM uses Equation (3) to calculate layer-weighted head, drawdown, and concentration values at multi-layer observation wells. The layer-weighted results are used for visualization in Line or Scatter charts.

$$h = \frac{\sum_{k=1}^n h_k \cdot w_k}{\sum_{k=1}^n w_k} \quad (3)$$

where n is the number of layers of the model; k is the layer number; h_k is the simulated head in layer k; and w_k is the weight of layer k. The layer weights do not need to add up to one. Single-layer and multi-layer head and drawdown values can be used by PEST to compare with the observed counterparts during a parameter estimation run. During a PEST calibration run, the layer weights can optionally be automatically adjusted based on updated HK values. For details, see **LAYERWEIGHTOPT** in Section 3.9.5.2.7. After a PEST calibration run, the layer weights can be updated based on the update HK values by using *More Actions* > *Apply PARVAL1* in the PEST Settings dialog box (Section 3.9.5.2).

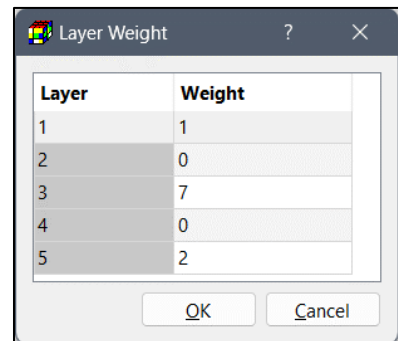


Figure 39. The Layer Weight dialog box

- **Elevation:** This column contains the ground surface elevations of the observation points.
- **Latitude and Longitude** (if the Geographic coordinate system is used) or **X and Y** (if the Cartesian coordinate system is used): These columns display the geographic coordinates (in WGS 84) or the user-defined Cartesian coordinates of the observation points.
- **XOffset and YOffset:** These are the easting and northing offsets (in meters) of the observation point label. These values can be used to adjust the display position of the label.
- **Notes:** A place for taking notes.

- **Add:** Click this button to add an observation point to the current layer.
- **Delete:** Click this button to delete the selected observation points (i.e., rows) from the table³¹.
- **Apply:** Click this button to apply and save the changes to the observation points and their data.
- **Pan To:** Click this button to center the current observation point in the middle of the Map Viewer.
- **Zoom To:** Click this button to zoom to the current observation point.
- **More Actions**
 - **Activate:** Select this option to turn on the selected observation points.
 - **Deactivate:** Select this option to turn off the selected observation points.
 - **Layer Weight:** Select this option to open the Layer Weight dialog box (Figure 39), and to assign the define Layer Weight values to the selected observation points.
 - **Clear:** Select this option to delete all observation points and their associated data.
- **Observations (right pane of the dialog box):** The controls of this pane act on the selected observation point (i.e., the selected row from the Observation Point table).
 - **Type:** This dropdown box is used to select the type of the observation data. The following data are supported.
 - **Hydraulic Head:** This is the height of the groundwater level above a reference point, often sea level or another fixed datum. Observed hydraulic head values of single-layer and multi-layer observation points can be used as calibration targets by PEST for model calibration and can be displayed in Scatter and Line Charts (see Sections 3.9.1.6.3 and 3.9.1.6.4).
 - **Drawdown:** Drawdown refers to the reduction in the groundwater level or hydraulic head since the start of the simulation. Observed drawdown values of single-layer and multi-layer observation points can be used as calibration targets by PEST for model calibration and can be displayed in Scatter and Line Charts.
 - **Concentration:** Observed and calculated concentration values of single-layer and multi-layer observation points can be displayed in Scatter and Line Charts.
 - **Streamflow:** This refers to measured or estimated stream discharge rates at stream gaging stations. Streamflow observations and simulated streamflow values can be displayed in Scatter and Line Charts. Streamflow observations can be used as calibration targets by PEST to compare with the simulated streamflow from the SFR2 Package for model calibration. See *GAGE* and *OBSPT* in Section 3.9.1.1.16 for details.
 - **Stage:** This refers to the measured or estimated water level or surface elevation at lake gaging stations. Stage observations and simulated stage values can be

³¹ You may select multiple rows from the table by using combinations of Ctrl+click and Shift+click.

displayed in Scatter and Line Charts. Stage observations can be used as calibration targets by PEST to compare with the simulated stage from the LAK2 Package for model calibration. See *GAGE* and *OBSPT* in Section 3.9.1.1.9 for details.

- **Compaction:** This refers to the measured or estimated compaction of a model layer since the start of the simulation. Observed and calculated concentration values of single-layer observation points can be displayed in Scatter and Line Charts. Multi-layer observations are not supported.
- **Subsidence:** This refers to the measured or estimated land subsidence since the start of the simulation. Subsidence values of single-layer observation points that are assigned to the first model layer can be displayed in Scatter and Line Charts. Multi-layer observation points and single-layer observation points that are assigned to layers other than the first layer are not supported.
- **Maximum Air Temperature:** This is intended to be used by GSFLOW alongside other meteorological data to simulate energy balance, snowmelt, and evapotranspiration. Daily maximum air temperature data over the simulation period of at least one observation point is required by GSFLOW. The unit is defined in *Models > GSFLOW > Settings*.
- **Minimum Air Temperature:** This is intended to be used by GSFLOW alongside other meteorological data to simulate energy balance, snowmelt, and evapotranspiration. Daily minimum air temperature data over the simulation period of at least one observation point is required by GSFLOW. The unit is defined in *Models > GSFLOW > Settings*.
- **Precipitation:** This is intended to be used by the Precipitation Distribution process of GSFLOW. Precipitation provides the primary source of water for surface runoff, infiltration, groundwater recharge, streamflow, and various watershed processes. Daily precipitation data over the simulation period of at least one observation point is required by GSFLOW. The unit is defined in *Models > GSFLOW > Settings*.
- **Humidity:** This is intended to be used by the *potet_pm_sta* module of the Potential Evapotranspiration (Section 3.9.4.11.2) process of GSFLOW. GSFLOW expects daily humidity data over the simulation period. The humidity values are expected to be given as [percentage] between 0 and 100.
- **Pan Evaporation:** This is intended to be used by the *potet_pan* module of the Potential Evapotranspiration (Section 3.9.4.11.2) process of GSFLOW. GSFLOW expects daily pan evaporation data over the simulation period. The unit is [inches].
- **Potential ET:** This is intended to be used by the *Climate-by-HRU* module of the Potential Evapotranspiration (Section 3.9.4.11.2) process of GSFLOW. In GSFLOW, Potential ET represents the maximum amount of water that could be evaporated and transpired by vegetation from a surface, assuming unlimited water availability. GSFLOW expects daily Potential ET data over the simulation period. The unit is [inches].

- **Wind Speed:** This is indented to be used by the *potet_pm_sta* module of Potential Evapotranspiration (Section 3.9.4.11.2) process of GSFLOW to calculate potential ET by using the Penman-Monteith formulation. GSFLOW expects daily wind speed data over the simulation period. The unit is [meters per second].
- **Snow Depth:** Daily snow depth data over the simulation period is an optional input. The unit for snow depth is [inches]. In GSFLOW, you typically do not need to enter snow depth time series. Instead, GSFLOW computes snow-related processes internally based on meteorological inputs and parameter settings related to snow accumulation and melt.
- **Solar Radiation:** This is indented to be used by the Solar Radiation process of GSFLOW. Solar radiation influences several key hydrological and energy balance processes, particularly evapotranspiration, snowmelt, and temperature dynamics. GSFLOW expects daily solar radiation data over the simulation period. The unit for solar radiation is [Langleys].
- **Species:** This dropdown box is activated only if the Type is Concentration and is used to select the species to which the observation data belong.
- **Table:** The table displays the observation data.
 - **Time:** Observation time, expressed in the simulation time (which is the elapsed time since the start of the simulation).
 - **Value:** Observed value of the selected Type.
 - **Weight:** The Weight of an observation gives a relative confidence level of the observed value. The higher the Weight value, the better is the measurement. The weight can be set at zero if needed (meaning that the observation takes no part in the calculation of the objective function during a parameter estimation process), but it must not be negative. Refer to the PEST User Manual (Doherty, 2016) for the function of weights in the parameter estimation process.
 - **Selected:** An observation value is selected as a calibration target if the corresponding box is checked.
 - **Date/Time:** This is the observation time expressed in calendar date/time. The values in this column are read-only.
- **Add:** Click this button to add a row to the Table.
- **Delete:** Click this button to delete the selected rows from the Table.
- **Target:** This dropdown box includes the following options.
 - **None:** Do not use any observation data (of the selected observation point) in model calibration, time-series charts, or scatter charts.
 - **All:** All observation data are selected as calibration targets and used in model calibration, time-series charts, and scatter charts.
 - **Selected:** The selected observation data are used as calibration targets in model calibration, time-series charts, and scatter charts.
- **More Actions:**

- **Select Target:** Select this option to open the Target Selection tool (Figure 40) that can be used to select a subset of observation data as calibration targets for model calibration and time series/scatter charts. The features of the Target Selection tool are given below.
- **Set Value:** Select this option to assign an observation value to the selected rows.
- **Set Weight:** Select this option to assign a weight value to the selected rows.
- **Clear Table:** Select this option to delete all observation data from the Table.
- **The Target Selection Tool** (Figure 40) can be used to select a subset of observation data. The features of this tool are as follows.
 - **The Time Series Chart** in the center of the tool displays the observation data time series. The vertical axis represents the measured property, such as hydraulic head. The horizontal axis represents the simulation time.
 - To pan the chart, move the mouse cursor into the chart, press and hold the left mouse button, and then move the mouse.
 - To zoom in/out on any part of the chart, place the mouse cursor within the chart, scroll the mouse wheel forward or backward.
 - You can limit the zoom and pan to the x or y direction by selecting the x or y-axis first. An axis is selected by clicking on it. A selected axis is colored in blue. To deselect an axis, simply click on any part of the respective chart.
 - **The features from left to right in top row of the tool include:**
 - Observation Point Name and Label is displayed on the upper-left corner of the dialog box.

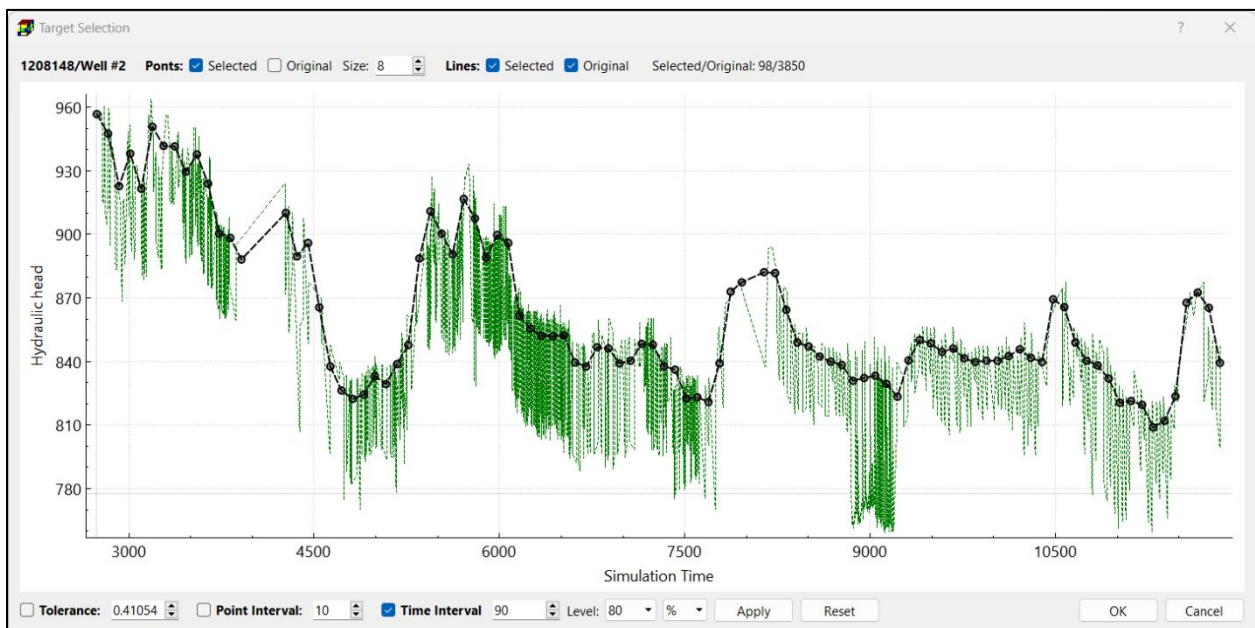


Figure 40. The Target Selection tool

- Points – Selected: Check the Selected box to display the selected observation data points as black circles in the Time Series chart. You can shift+click on a black circle to remove it. You can ctrl+click anywhere on the chart to add black circles.
 - Points – Original: Check the Original box to display the original observation data points as translucent green circles.
 - Points – Size: this defines the size in pixels for the black and green circles.
 - Lines – Selected: Check the Selected box to display black dashed lines connecting the selected observation data points.
 - Lines – Original: Check the Original box to display green dashed lines connecting the original observation data points.
 - Selected/Original: this label shows the numbers of the selected and original data points.
- **The features from left to right in bottom row of the tool include:**
- Tolerance: If this box is checked, the Ramer–Douglas–Peucker algorithm is used to find a similar time series with fewer points. The value of Tolerance is used to control how close the original and the simplified time series should be.
 - Point Interval: If this box is checked, the original data points at the given point interval will be selected.
 - Time Interval: The time interval is expressed in the time unit of the model. If this box is checked, the mid-point of each time interval that contains observational data will be set as a calibration target.
 - Level: The settings of the Level dropdown box and the Option dropdown box to the right determine the value of the mid-point of a time interval.
 - If Level = Auto, the mid-point of a time interval will be set as a calibration target, with the value equal to the average observed value within the time interval.
 - If Level ≠ Auto and Option = pth, the mid-point of a time interval will be set as a calibration target, with the value equal to the Level percentile of the observed values within the time interval.
 - If Level ≠ Auto and Option = %, the mid-point of a time interval will be set as a calibration target, with the value = $\text{min} + \text{Level}/100 * (\text{max} - \text{min})$, where min and max are respectively the minimum and maximum observation values within the time interval.
 - Apply: Click this button to apply changes to the settings.
 - Reset: Click this button to reset the features of the chart to their default settings.
 - OK: Close the tool and store the settings.

5. Export/Import observation points:


- **Export to Shapefile:** Right click on the Observation Points group on the TOC, select Export to Shapefile from the popup menu, and then select a shapefile in the Save As dialog box.
- **Export to Excel File:** Right click on the Observation Points group on the TOC, select Export to Excel File from the popup menu, and then select an Excel file in the Save As dialog box. The Excel file contains the observation points and observation data. Its format is described in Section 5.1.9.
- **Import and Replace:** Right click on the Observation Points group on the TOC, select *Import and Replace* from the popup menu, and then select Excel files in the Open File dialog box. The existing observation points and their data will be deleted, and then the data of the selected Excel files will be imported. The Excel files must be saved in the format described in Section 5.1.9. If you need a template file, use the Export to Excel File function to create a template file.
- **Import and Append:** Right click on the Observation Points group on the TOC, select *Import and Append* from the popup menu, and then select Excel files in the Open File dialog box. The data of the selected Excel files will be imported and append to the existing data. The Excel files must be saved in the format described in Section 5.1.9. If you need a template file, use the Export to Excel File function to create a template file.

3.6.4. MODPATH Tool

The MODPATH Tool may be accessed only when the active layer is a MODPATH instance. This tool is used to define starting locations for particle tracking. For details of running MODPATH, see Section 4.2.4.

3.6.5. Measure Tool

The Measure tool can be used to measure length of a line or a path. To use this tool, follow the steps provided below.

1. Open the Measure Tool (Figure 41) by clicking the  button on the toolbar or selecting Measure Tool from the View menu of the Main Window (Figure 1).
2. Position the portion of the map you want to measure within the Map Viewer.
3. Select the Line or Path tab in the Measure tool and click in the Map Viewer to set the beginning point for the line or path. A line consists of two vertices, while a path can consist of two or more vertices. When using the path function, each additional click adds a new vertex. Vertices are displayed as red squares. The length

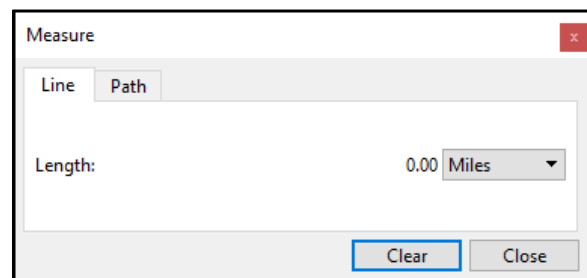




Figure 41. The Measure Tool

(distance) of the line or path is calculated as the great-circle distance using latitude/longitude coordinates from point to point.

4. Once a line or a path is defined, its shape can be changed by moving its vertices. To move the vertices of a line or path, follow these steps:
 - Activate the defined line or path by selecting the Line or Path tab in the Measure tool. Only one line or path can be made at a time.
 - Mouse over the desired vertex. When the mouse cursor turns into a , click and hold the left button down, move the mouse to drag the vertex to a new position, and release the left button.
 - The length of the line or path is updated while its vertex is being dragged.
5. Click Clear to remove the line and path.
6. To close the Measure tool, click the Close button or click the  button on the toolbar.

3.6.6. Row Cross Section Tool

The Row Cross Section Tool (Figure 42) displays the model grid, model features (such as STR or WEL cells), and model results (such as contours, flow vectors and/or pathlines) along a selected model row. The row cross section displays the model columns from the left to the right. The first model column is displayed on the left-hand side of the cross section and the last model column is on the right-hand side. The features of this tool are described as follows.

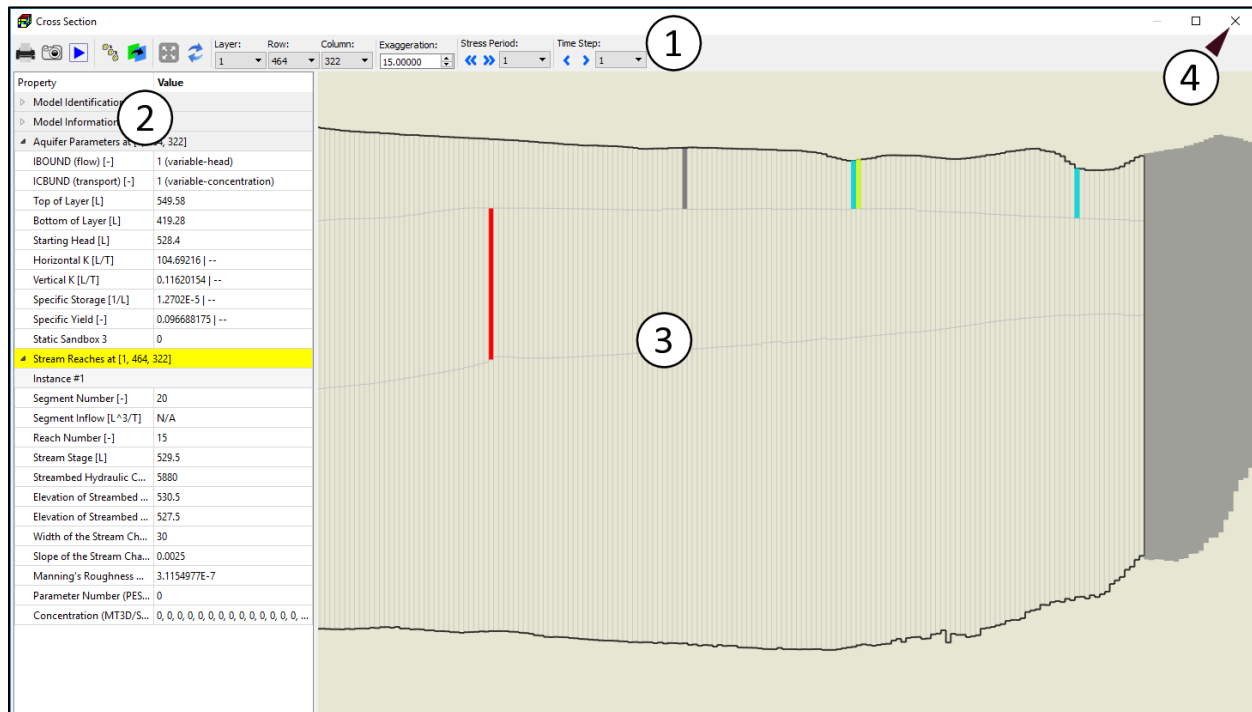






Figure 42. The Row Cross Section Tool


1. The **Toolbar** contains several buttons and controls as given below.


 **Print Image.** Click this button to open the Print dialog box that allows you to print the contents of the cross-section window. See Section 3.5.11 for details.


 **Save Image.** Click this button to save the contents of the cross-section window as an image file. See Section 3.5.9 for details.

 **Animation.** Click this button to open the Animation dialog box that allows you to run animations with the model data and optionally save animations into video files or sequences of image files. See Section 3.5.93.5.10 for details.

 **Turn on/off the *Copy Cell Data* function.** If this button is depressed, the cell value(s) of the current cell will be copied to all cells passed by the grid cursor.

 **Turn on/off the *Copy Slice Data* function.** If you move to another row while this button is depressed, the cell values of the current model row will be copied to the destination row.

 **Zoom out to the entire grid.** Click this button to fit the model to the cross-section window.

 **Remove highlight.** Within the data editor (Section 3.4.2), you can double-click on a cell to highlight all other cells that have the same value as the clicked cell. The highlight mask may be set to be cleared automatically (see Preferences in Section 3.5.12). Click this button to clear the highlight mask if it is not cleared automatically.

- **Layer/Row/Column:** These dropdown boxes show the numbers of layer, row, and column of the cell pointed to by the grid cursor. You may use these dropdown boxes to move the grid cursor.
- **Exaggeration:** This is the vertical exaggeration factor that is used to adjust the appearance height of the model cross-section.
- **Stress Period:** The dropdown box displays the current stress period. You may use the dropdown box to move to another period. Alternatively, click the **◀** or **▶** button to move to the previous or next stress period, respectively.
- **Time Step:** The dropdown box displays the current time step within the selected Stress Period. You may use the dropdown box to move to another time step. Alternatively, click the **◀** or **▶** button to move to the previous or next time step, respectively.

2. **The Property Window:** This window displays the meta data of the model. If model results are displayed or a model data set is being edited, the properties of the cell pointed to by the *grid cursor* are displayed in this window. The cell data that is currently being edited is highlighted in yellow.

3. **The Cross-Section Window:** This window displays the model grid and model cell features along the model row specified by the Row dropdown box.


- **Display Settings:** The cross-section window shares the same display settings with the Map Viewer. For example, if contours are displayed on the Map Viewer, they will also be displayed on the cross-section window. See Chapter 4 for details of available display settings.

- **Move the cross-section view in any direction:** Place the mouse cursor within the window and then drag to move the view. Alternatively, the w, e, d, c, x, z, a, and q keys can be used to move the view towards the north, northeast, east, southeast, south, southwest, west, and northwest, respectively.
 - **Move the grid cursor.** There are several ways to move the grid cursor.
 - Move the grid cursor to a cell by left-clicking on that cell.
 - Drag the grid cursor with the mouse.
 - Use the arrow keys to move horizontally. Use the Shift+PageUp and Shift+PageDown keys to move to the adjacent rows.
 - Use the Layer/Row/Column dropdown boxes on the toolbar.
 - **Zoom in:** Place the mouse cursor within the window, press and hold the right button, and move the mouse forward. Release the button when you reach the desired zoom level. Alternatively, if your mouse has a scroll wheel, it can be used to zoom in by scrolling it forward.
 - **Zoom out:** Place the mouse cursor within the window, press and hold the right mouse button, and move the mouse backward. Release the button when the desired zoom level is reached. Alternatively, if your mouse has a scroll wheel, it can be used to zoom out by scrolling it backward.
 - **Edit Cell data:** If you invoke the Row Cross-Section Tool while editing a certain model data, you may edit the cell data by right-clicking on the cell of interest. You can double-click on a cell (or press the Insert key) to highlight all other cells that have the same cell value as the clicked cell.
4. **Close the Tool:** To close the tool, click the [X] button located on its upper-right corner. After closing this tool, the grid cursor on the Map Viewer will be moved to the cell pointed to by the Layer/Row/Column dropdown boxes of this tool. The model on the Map Viewer will be redrawn if necessary.


3.6.7. Column Cross Section Tool

The Column Cross Section Tool displays the model grid, model features (such as STR or WEL cells), and model results (such as contours, flow vectors and/or pathlines) along a selected model column. The column cross section displays the model rows from the left to the right. The first model row is displayed on the left-hand side of the cross section and the last model row is on the right-hand side. See previous section for details about the features of this tool.


3.6.8. Pin Tool



Select *Tools > Pin Tool* or push down the  button on the toolbar to disable map pan and zoom with the mouse.

3.6.9. Rotation Tool

Select *Tools > Rotation Tool* or push down the  button on the toolbar to orientate the display of the active model grid horizontally. All other visual objects, including other models, will be rotated with the active model grid accordingly.

3.6.10. Location Tool

Select *Tools > Location Tool* or click the  button on the toolbar to open the Locator dialog box that has two tabs. The Model Cell tab (Figure 43) allows you to locate a model cell, and the Location tab (Figure 44) allows you to pinpoint a location.

1. To locate a model cell
 - Click on the Model Cell tab (Figure 43),
 - Enter the row and column of the cell,
 - Click OK to zoom to the model cell. A cross  will be placed in the center of the cell for a short time.
2. To pinpoint a location
 - Click the Location tab (Figure 44),
 - Specify the Projection, Datum, and Coordinates of the desired location,
 - Click OK to zoom to the location. A cross  will be placed in the given location for a short time.

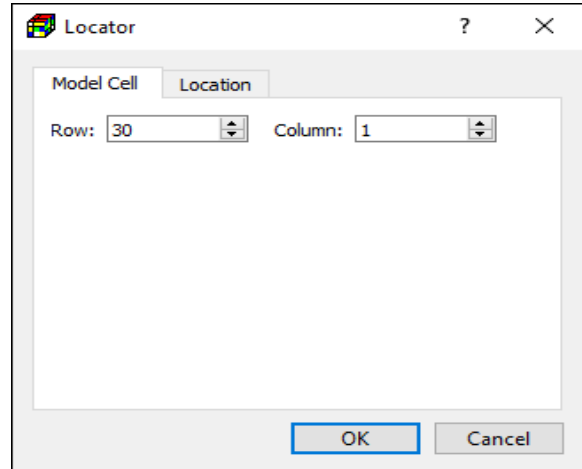


Figure 43. The Model Cell tab of the Locator dialog box

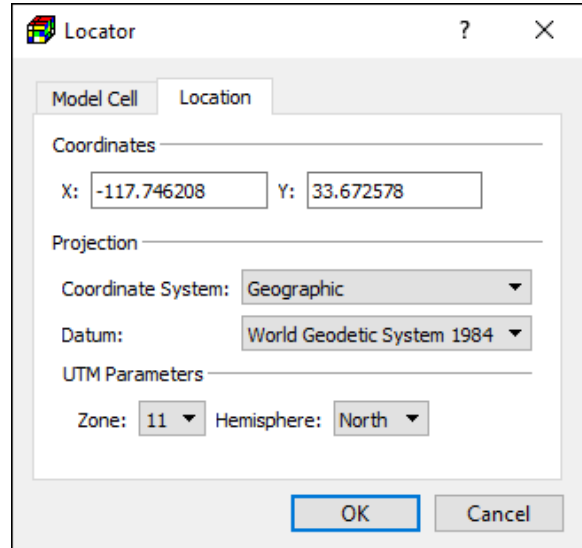




Figure 44. The Location tab of the Locator dialog box


3.6.11. Finish Edit

Click on the  button on the toolbar to finishing editing the grid or the model data. If changes have been made, you will be prompted to save or discard the changes.


3.6.12. Cell-by-Cell Tool

Within the Data Editor, you may select *Tools > Cell-by-Cell Tool* or push down the  button on the toolbar to switch to the Cell-by-Cell input method. See section 3.4.2.1 for details.


3.6.13. Polyline Tool

Within the Data Editor, you may select *Tools > Polyline Tool* or push down the  button on the toolbar to switch to the Polyline input method. See section 3.4.2.2 for details.


3.6.14. Polygon Tool

Within the Data Editor, you may select *Tools > Polygon Tool* or push down the  button on the toolbar to switch to the Polygon input method. See section 3.4.2.3 for details.


3.6.15. Pilot Points Tool

Within the Data Editor, you may select *Tools > Pilot Points Tool* or push down the  button on the toolbar to switch to the *Pilot Points* input method. See section 3.4.2.4 for details.


3.6.16. Cell Copy

Within the Data Editor, you may select *Tools > Cell Copy* or push down the  button on the toolbar to turn on the *Copy Cell Data* function. If this button is depressed, the cell value(s) of the current cell will be copied to all cells passed by the grid cursor.


3.6.17. Layer Copy

Within the Data Editor, you may select *Tools > Layer Copy* or push down the  button on the toolbar to turn on the *Copy Layer Data* function. If you move to another layer while this button is depressed, the cell values, polylines, and polygons of the current layer will be copied to the destination layer.

3.6.18. Grid Size Copy

Within the Grid Editor, you may select *Tools > Grid Size Copy* or push down the  button on the toolbar to turn on the *Copy Grid Size* function. If this button is depressed, the column and row width of the current cell will be copied to all cells passed by the grid cursor.

3.6.19. Grid Split Copy

Within the Grid Editor, you may select *Tools > Grid Split Copy* or push down the  button on the toolbar to turn on the *Copy Grid Split Factor* function. If this button is depressed, the column and row split factors of the current cell will be copied to all cells passed by the grid cursor.

3.6.20. Tools > Data Menu

Within the Data Editor, you may use this menu to manipulate cell-by-cell data of the current layer and/or the entire model. A menu item is dimmed if it is not supported.

1. **Tools > Data > Matrix:** Select this menu item to open the Matrix dialog box that displays cell values in a table. This function is available only when editing IBOUND, ICBUND, Layer Top/Bottom elevations, aquifer parameters, concentration values, and certain flow packages with matrix-based input data, such as Evapotranspiration or Recharge packages. The features of the Matrix dialog box are as follows.

- **Table:** The table displays cell values of the selected parameter. The columns and rows of the table correspond to the model columns and rows. You may edit the value of a cell by clicking on the cell and then typing in the desired value. You may edit the values of a group of cells by selecting those cells with the mouse, and then typing in the desired value.
 - **Parameter:** The cell values of the selected parameter are displayed in the table.
 - **Column Width:** This defines the display width of the columns of the table.
 - **Statistics:** This line contains the statistics of the displayed values.
2. **Tools > Data > Reset:** Select this menu item to open the Reset dialog box, which is used to assign uniform cell values to the current model layer or to the entire model.

The options *Apply to the entire model* and *Apply to the current layer* are available when editing IBOUND, ICBUND, layer top/bottom elevations, aquifer parameters, or concentration values.

- **Apply to the entire model:** If this option is selected, the specified value(s) in the Reset Matrix dialog box will be applied to all cells of the entire model.
- **Apply to the current layer:** If this option is selected, the specified value(s) in the Reset Matrix dialog box will be applied to all cells of the current model layer.

3. **Tools > Data > Calculate**

- **Sum:** Calculate the sum of cell values for each vertical column of cells and put the results in the current layer.
- **Average:** Calculate the average cell values for each vertical column of cells and put the results in the current layer.

4. **Tools > Data > Modify:** Open the Modify Data dialog box (Figure 45) that allows you to modify cell values.

- **Parameter:** The modification operation applies to the selected parameter.
- **Filters:** The modification operation applies to the cells that meet the criteria defined in the Filters group.

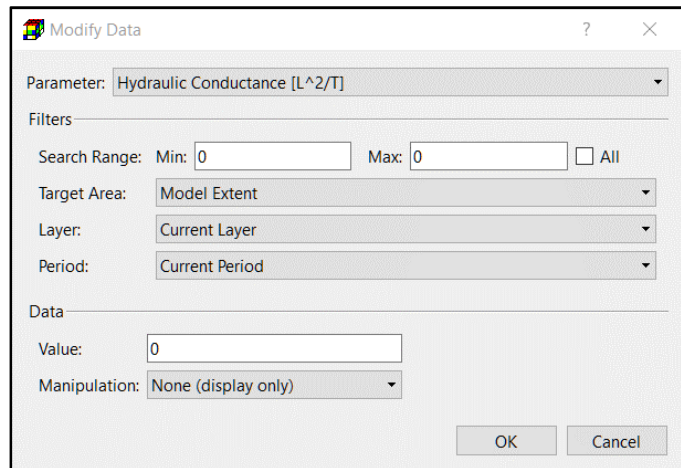


Figure 45. The Modify Data dialog box

- **Search Range:** this criterion is met if *All* is checked or if the selected parameter value of a cell lies within the given *Min* and *Max* values.
- **Active Cells Only:** this criterion is met if a cell is a variable-head cell or a constant head cell, i.e., cells with IBOUND \neq 0. This criterion is ignored when editing IBOUND.


- **Target Area:** this criterion is met if a cell lies within the selected target area that may include Model Extent, Views (Section 2.1), and Polygons (Section 3.4.2.3).
- **Layer:** This dropdown box contains *Current Layer* and *All Layers*. If *Current Layer* is selected, cells in the current layer meet the criterion. If *All Layers* is selected, all cells of the model meet the criterion.
- **Period:** This dropdown box contains *Current Period* and *All Periods*. If *Current Period* is selected, the modification operation applies to the current stress period. If *All Periods* is selected, the modification operation applies to all stress periods. This dropdown box is activated only when editing packages³² that support this feature.

For example, if *Target Area* is a polygon, *Layer* is set to *All Layers* and *Active Cells Only* is checked, the modification operation will apply to active cells (of all layers) that lie within the polygon and have values within the *Search Range*.

- **Value:** The user-specified value that is used to modify the cell values.
- **Manipulation:** The final cell values are obtained by cell-by-cell calculation according to the selected manipulation rule as follows.
 - **None (display only):** No modification takes place.
 - **Replace:** Final cell value = (user-specified value)
 - **Add:** Final cell value = (current cell value) + (user-specified value)
 - **Subtract:** Final cell value = (current cell value) – (user-specified value)
 - **Multiply:** Final cell value = (current cell value) × (user-specified value)
 - **Divide:**
 - If the user-specified value $\neq 0$:
Final cell value = (current cell value) ÷ (user-specified value)
 - If the user-specified value = 0, the current cell value remains unchanged.
 - **Minimum:** Final cell value = minimum(current cell value, user-specified value)
 - **Maximum:** Final cell value = maximum(current cell value, user-specified value)
 - **Average:** Final cell value = average(current cell value, user-specified value)
 - **Power:** Final cell value = (current cell value) ^ (user-specified value)

³² The supported packages are CHD, DRN, DRT, GHB, RIV, MNW2, and WEL.

5. **Tools > Data > Import:** Open the Import Data dialog box (Figure 46) that allows you to import cell values from several types of files.

- **File:** Click the  button to select a file to be imported. After selecting a file, PM will attempt to determine its file type. If a supported file type is determined, the *File Type* dropdown box (see below) will be updated accordingly.
- **File Type:** The following file types are supported when editing IBOUND, ICBUND, and Layer Top/Bottom elevations, aquifer parameters, concentration values, and certain flow packages with matrix-based input data, such as Evapotranspiration or Recharge package. Refer to Section 5.1 for the formats of these file types.

- **2D Matrix**
- **2D Modflow Array**
- **3D Modflow Array**
- **SURFER ASCII GRD File**³³
- **Comma-Separated Values File**³⁴
- **ASCII Raster**³⁵
- **Shapefile**³⁶ as defined by (ESRI, 1998).

For packages with list-based input data³⁷, the following file types are supported. If the file types 2D Matrix or 2D Modflow Array are used, the imported data only apply to existing instances of package-specific features (e.g., existing river or drain cells), except the manipulation option *Create* (see below).

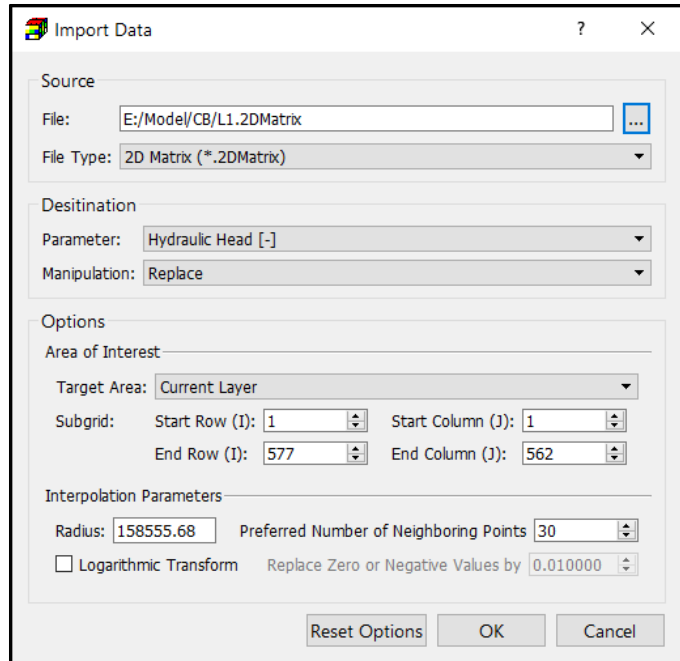


Figure 46. The Import Data dialog box

³³ Only models with uniform column width and row width are supported. The GRD files must be saved in the Surfer ASCII GRD file format.

³⁴ Input values are interpolated to the model cells using the Interpolation Parameters.

³⁵ A projection file⁷ is required for geo-referencing of an ASCII Raster file.

³⁶ This filetype is supported only if the Geographic coordinate system is used. Only shapefiles with polygon or polyline features are supported. Before importing a selected shapefile, you need to select a column field from the shapefile attribute table. The selected attribute of a polygon will be imported to a cell, if its center lies within the polygon. The selected attribute of polyline vertices will be used for interpolation to cells.

³⁷ These packages use list-based input data: CHD, DRN, DRT1, FHB1, GHB, HFB6, MNW1, RES, RIV, STR, SFR2, and WEL.

- **Package-specific format**³⁸
- **2D Matrix**
- **2D Modflow Array**
- **Destination**
 - **Parameter:** The import operation will be applied to the selected parameter.
 - **Manipulation:** The final imported cell values are obtained by cell-by-cell calculation according to the selected manipulation rule as follows.
 - **Replace:** Final cell value = imported cell value.
 - **Add:** Final cell value = (current cell value) + (imported cell value)
 - **Subtract:** Final cell value = (current cell value) – (imported cell value)
 - **Multiply:** Final cell value = (current cell value) × (imported cell value)
 - **Divide:**
 - If the imported cell value $\neq 0$:
Final cell value = (current cell value) ÷ (imported cell value)
 - If the imported cell value = 0, the current cell value remains unchanged.
 - **Minimum:** Final cell value = minimum(current cell value, imported cell value)
 - **Maximum:** Final cell value = maximum(current cell value, imported cell value)
 - **Average:** Final cell value = average(current cell value, imported cell value)
 - **Power:** Final cell value = (current cell value) ^ (imported cell value)
 - **Create** (*supported by CHD, DRN, DRT1, GHB, RES, RIV, and WEL packages*):
When importing a matrix, package-specific features (such as river or drain cells) are created at cells with non-zero imported values. The imported values are then assigned to the selected *Parameter* of these newly created instances.
- **Options:** If an option is applicable, it is activated; otherwise, it is dimmed.
 - **Area of Interest.**
 - **Target Area and Subgrid:** The area of interest is defined by the intersection of the *Target Area* and the *Subgrid*. The data will be imported to the area of interest based on the following rules.
 - If a 2D Matrix, 2D Modflow Array, 3D Modflow Array, or SURFER ASCII GRD file is used, the value located at [i, j] of the matrix will be imported to the cell located at [i + StartRow - 1, j + StartColumn - 1] and inside the *Area of Interest*.
 - If a Comma-Separated Values File or an ASCII Raster is use, the input values will be interpolated to the cells of the Subgrid. The resulting matrix with the interpolated values will then be imported in the same way as described above.
 - **Active Cells Only:** if this is checked, the imported data will only be applied to variable-head or constant head cells.

³⁸ Files in a package-specific format are created by using Tools > Data > Export.

- **Interpolation Parameters:** These parameters are used to interpolate the input values from the Comma-Separated Values file, the ASCII Raster file, or the Shapefile with polyline, point or multipoint features to the model grid. The interpolation process is based on a weighted method. In this method, the grid value C_{ij} at the cell [j, i] is the weighted average of surrounding data values, i.e.,

$$C_{ij} = \sum_{k=1}^n C_k W_k(r) / \sum_{k=1}^n W_k(r) \quad (4)$$

where C_k is the measured value at the k^{th} measuring point, n the number of surrounding data values, $W_k(r)$ is the weighting function, and r the distance from the grid point to the measuring point. The key component of a weighted method is the weighting function. A survey of weighting functions is presented by (Goodin, et al., 1979). In PM, the weighting function developed by Shepard (1968) is used, which improves the pure inverse distance weighting method by adding functions to dynamically determine (1) the radius of influence based on data density and distribution, and (2) the weighting factors based on distance between model cells and data points in the interpolation algorithm.

- **Radius and Preferred Number of Neighboring Points (PNP):** These values determine the neighboring data points that should be used to estimate the value at that cell with the following rules:
 - The number of points within the Radius (NPR) of the grid cell is determined by PM with a fast search algorithm (Mount, et al., 2010).
 - The number of neighboring points (NP) used by the Shepard’s method to estimate the value at the grid cell is determined as follows:

$$NP = MinNP \quad \text{if } NPR < MinNP,$$

$$NP = NPR \quad \text{if } MinNP < NPR < MaxNP, \text{ and}$$

$$NP = MaxNP \quad \text{if } NPR > MaxNP,$$
 where $MinNP = PNP \times 2/3$ and $MaxNP = PNP \times 4/3$
 - The radius used by the Shepard’s method is the radius that encompasses those NP points next to the grid cell.
- **Logarithmic Transform:** Check this box to log-transform the input values prior to performing interpolation. This is typically necessary if the input data is positively skewed, such as concentration measurements. Without log-transformation, the estimates based on positively skewed data sets could be significantly higher (over-estimation) because of the impact of high values.
- **Replace Zero or Negative Values by:** Prior to log-transform, the input values that are smaller than or equal to zero are replaced by the value defined here.

6. **Tools > Data > Copy:** This menu item opens the Copy Data dialog box (Figure 47) that allows you to copy cell values from the selected source to a destination.

- **Source:** This group defines the source data that may be copied to destination.

- **Parameter:** This dropdown box contains a list of static parameters (e.g., layer top elevation, layer bottom elevation, IBOUND, ICBUND, HK, VK, etc.).

- **Layer:** This dropdown box contains a list of layer numbers and an entry called *Entire Model*.

- If a layer number is selected and the *Layer* dropdown box of the *Destination* group is set to *Current Layer*, the data of the selected layer will be copied to the current layer of the destination parameter.
- If a layer number is selected and the *Layer* dropdown box of the *Destination* group is set to *Entire Model*, the data of the selected layer will be copied to every layer of the destination parameter.
- If *Entire Model* is selected, the *Layer* dropdown box of the *Destination* group will also be set to *Entire Model*, and the source parameter of the entire model will be copied to the destination parameter.

- **Destination:** *Parameter* and *Layer* define the destination where the source data will be copied to.

- **Parameter:** the source data will be copied to the selected parameter. For packages with list-based input data³⁷, the data only apply to existing instances of package-specific features, such as wells or drains.

- **Layer:** this dropdown box contains *Current Layer* and *Entire Model*. See *Layer* in the Source group above for details.

- **Target Area:** This may include *Model Extent*, *Views* (Section 2.1), and *Polygons* (Section 3.4.2.3). Source data will only be copied to the cells within the selected target area.

- **Manipulation:** The final cell values are combined cell-by-cell with existing values of the destination parameter according to the manipulation rules listed below. In these equations, current value represents the existing cell value of the destination parameter, and source value represents the imported cell value:

- **Replace:** Final cell value = source value.
- **Add:** Final cell value = (current value) + (source value).
- **Subtract:** Final cell value = (current value) – (source value).

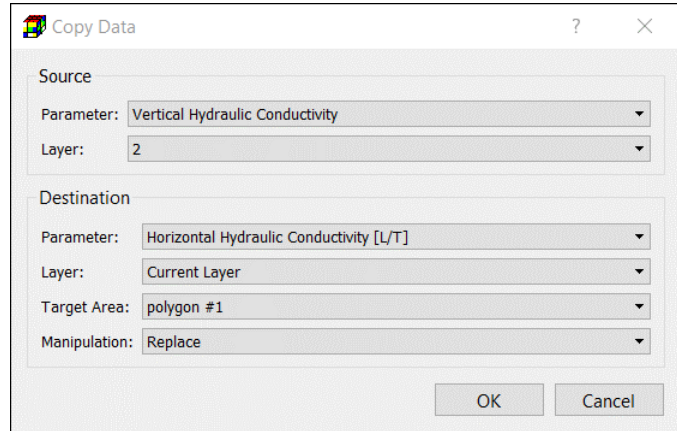



Figure 47. The Copy Data dialog box

- **Multiply:** Final cell value = (current value) × (source value).
- **Divide:**
 - If the imported cell value ≠ 0:
Final cell value = (current value) ÷ (source value)
 - If the source value = 0, the current value remains unchanged.
- **Minimum:** Final cell value = minimum(current value, source value).
- **Maximum:** Final cell value = maximum(current value, source value).
- **Average:** Final cell value = average(current value, source value).
- **Power:** Final cell value = (current value) ^ (source value).

7. **Tools > Data > Export:** This menu item opens the Export Data dialog box (Figure 48).

- **File:** Click the  button to select a file to be exported.
- **Parameter:** The data of the selected parameter will be exported.
- **Save As:** The exported file type. The following file types (refer to Section 5.1 for their formats) are supported when editing IBOUND, ICBUND, and Layer

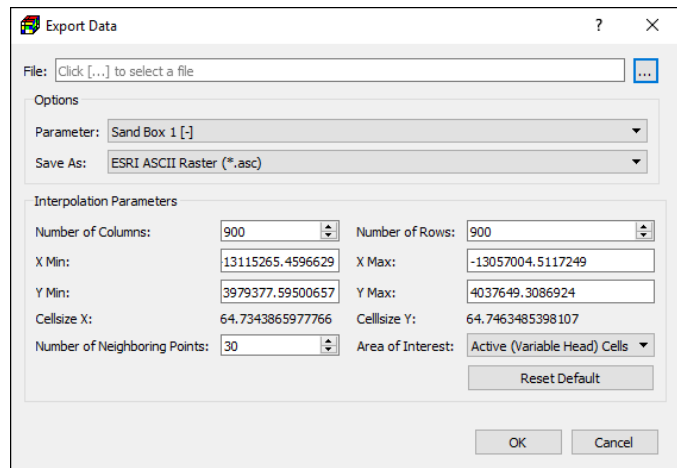


Figure 48. The Export Data dialog box

Top/Bottom elevations, aquifer parameters, concentration values, and certain flow packages with matrix-based input data, such as Evapotranspiration or Recharge package. For packages with list-based input data³⁷, package-specific formats are supported instead.

- **2D Matrix**³⁹
- **2D Weight Matrix**³⁹
- **2D Modflow Array**
- **3D Modflow Array**
- **SURFER ASCII GRD File**³³
- **Comma-Separated Values File**⁴⁰
- **ASCII Raster**⁴¹
- **Interpolation Parameters:** This group is enabled only if the selected file type is ASCII Raster. When exporting to this format, the cell-by-cell values of the selected

³⁹ 2D Matrix and 2D Weight Matrix share the same format. In the latter, the cell values are divided by the sum of cell values in the layer before exporting. If the sum of cell values is zero, it is replaced by 1.

⁴⁰ The coordinates of the exported files are expressed in [longitude, latitude], where longitude and latitude are expressed in WGS84.

⁴¹ The Mercator coordinates will be used in the exported raster file. A projection file⁷ will be created along with the raster file.

parameter are interpolated to a raster defined by the parameters of this group. The cell values equal to Nodata_Value (-1E20) are excluded from the interpolation. The interpolation is based on the Shepard's method (see above).

- **Number of Columns** and **Number of Rows**: The exported raster will have the defined numbers of columns and rows.
- **X Min, X Max, Y Min, and Y Max**: These are respectively the minimum and maximum X and Y coordinates (expressed in the Mercator coordinates if the Geographic coordinate system is used) of the raster. The default values cover the entire model grid.
- **Cellsize X** and **Cellsize Y**: These are respectively the column and row widths of the raster. These values are calculated based on the input values given above.
- **Number of neighboring points**: This defines the number of neighboring data points that should be used to estimate the value at a raster cell.
- **Exclude Inactive (no-flow) Cells**: If this box is checked, values of inactive cells are not used for the interpolation.
- **Area of Interest**: Cell-by-cell values are interpolated to the raster points within the selected area of interest and Nodata_Value (-1E20) is assigned to all other raster points. The results of the options "All" and "Model Bounding Box" are the same if the model grid is not rotated (or one of the model grid axes is parallel to a meridian).
- **Reset Default**: Click this button to reset the values of numbers of columns and rows, as well as X Min, X Max, Y Min, and Y Max to their default settings.

3.6.21. Tools > Package Menu

This menu provides the following package-specific functions.

1. **Tools > Package > Clear Package**: Clear all data of the package that is being edited.
2. **Tools > Package > Replace Package (via Name File)**: When editing one of the following MODFLOW packages, you can replace an existing package with one from another model by selecting its corresponding Name file in the Open File dialog box. The imported package file must be formatted according to the specifications described in the MODFLOW user guide (Harbaugh, 2005) or in the package-specific documentation.
 - Drain (DRN) Package
 - Drain with Return Flow (DRT1) Package
 - Evapotranspiration (EVT) Package
 - Evapotranspiration Segments (ETS1) Package
 - Flow and Head Boundary (FHB1) Package
 - General-Head Boundary (GHB) Package
 - Horizontal-Flow Barrier (HFB6) Package
 - Interbed-Storage (IBS) Package
 - Lake (LAK3) Package
 - Multi-Node Well (MNW1) Package

- Multi-Node Well 2 (MNW2) Package
- Recharge (RCH) Package
- Reservoir (RES1) Package
- River (RIV) Package
- Stream (STR) Package
- Streamflow-Routing (SFR2) Package²⁴
- Subsidence and Aquifer Compaction (SUB) Package
- Seawater Intrusion 2 (SWI2) Package
- Time Variant Specified Head (CHD) Package
- Unsaturated-Flow (UZF1) Package
- Well (WEL) Package

3. **Tools > Package > Replace Package (via Single File):** You can import a package by selecting its input file in the Open File dialog box. Ensure that the correct file format (e.g., MODFLOW-2005 Free Format or MODFLOW-2005 Fixed Format) is selected:

- Choose MODFLOW-2005 Free Format if the input file uses free-format notation.
- If the input file was originally created using MODFLOW's fixed-format notation, select MODFLOW-2005 Fixed Format.

Tip: If a MODFLOW Name file is available, it is recommended to use Replace Package (via Name File) instead, as it ensures the correct file format is automatically recognized.

4. **Tools > Package > Append Package (via Name File)** and **Tools > Package > Append Package (via Single File):** When editing any of the following MODFLOW packages, you can import data from package input files and append it to the existing data. The imported input files must follow the formatting guidelines described in the MODFLOW user guide (Harbaugh, 2005) or the specific documentation for each package.

- Drain (DRN) Package
- Drain with Return Flow (DRT1) Package
- General-Head Boundary (GHB) Package
- Horizontal-Flow Barrier (HFB6) Package
- Multi-Node Well 2 (MNW2) Package
- River (RIV) Package
- Time Variant Specified Head (CHD) Package
- Well (WEL) Package

5. **Tools > Package > Append Wells:** When editing the WEL or MNW2 packages, you can import and append wells from external Well Files. These files store coordinates, well screen intervals, pumping/injection rates per stress period, and optionally, concentration data for injected fluids. The format of these Well Files is described in Section 5.1.18 and differs from the standard MODFLOW input-file format.

6. **Tools > Package > Auto Layer Number:** When editing any of the packages listed below, this option allows you to automatically move package-specific features (e.g., drains, wells) vertically to the appropriate model layer based on their elevation values. Features are

reassigned to the layer corresponding to their elevation, ensuring proper placement within the model grid.

- Drain (DRN) Package: drain elevation.
- Drain with Return Flow (DRT1) Package: drain elevation.
- General-Head Boundary (GHB) Package: hydraulic head on boundary.
- Reservoir (RES1) Package: land surface elevation (i.e., elevation of reservoir bed)
- River (RIV) Package: elevation of the riverbed bottom.
- Stream (STR) Package: elevation of the streambed bottom.
- Streamflow-Routing (SFR2) Package: top elevation of the streambed – thickness of the streambed.

A package feature is reassigned to the appropriate model layer within the same vertical column of cells based on the following rules:

- If the elevation of a feature is higher than the top elevation of the highest active cell, the feature is moved to that cell.
- If the elevation of a feature is lower than the bottom elevation of the lowest active cell, the feature is moved to that cell.
- If the elevation of a feature lies between top and bottom elevations of an active cell, the feature is moved to that cell.
- If none of above applies, the feature is not moved.

3.7. Discretization Menu

3.7.1. Time

The menu item opens the Time dialog box (Figure 49) that defines the temporal discretization scheme of the model.

1. **Time Unit:** This dropdown box defines the time unit of the model data and stress period length.
2. **Auto Update Period Length:** If this box is checked, the stress period length will be updated whenever the time unit is changed. The values of all other model parameters are not affected. For conversion purposes, a year has 365.2425 days.
3. **Date/Time Format:** This controls the display format of the columns *Start D/T* and *End D/T* of the table.
4. **Table:** MODFLOW divides the total time length of a simulation into stress periods, which are, in turn, divided into time steps. The table displays the temporal discretization data.
 - **Period** is the stress period number.
 - **Steady State**⁴²: Check the box, if a period is steady state; uncheck the box if transient.

⁴² If the Interbed Storage (IBS) Package is used, only the first stress period may be steady state.

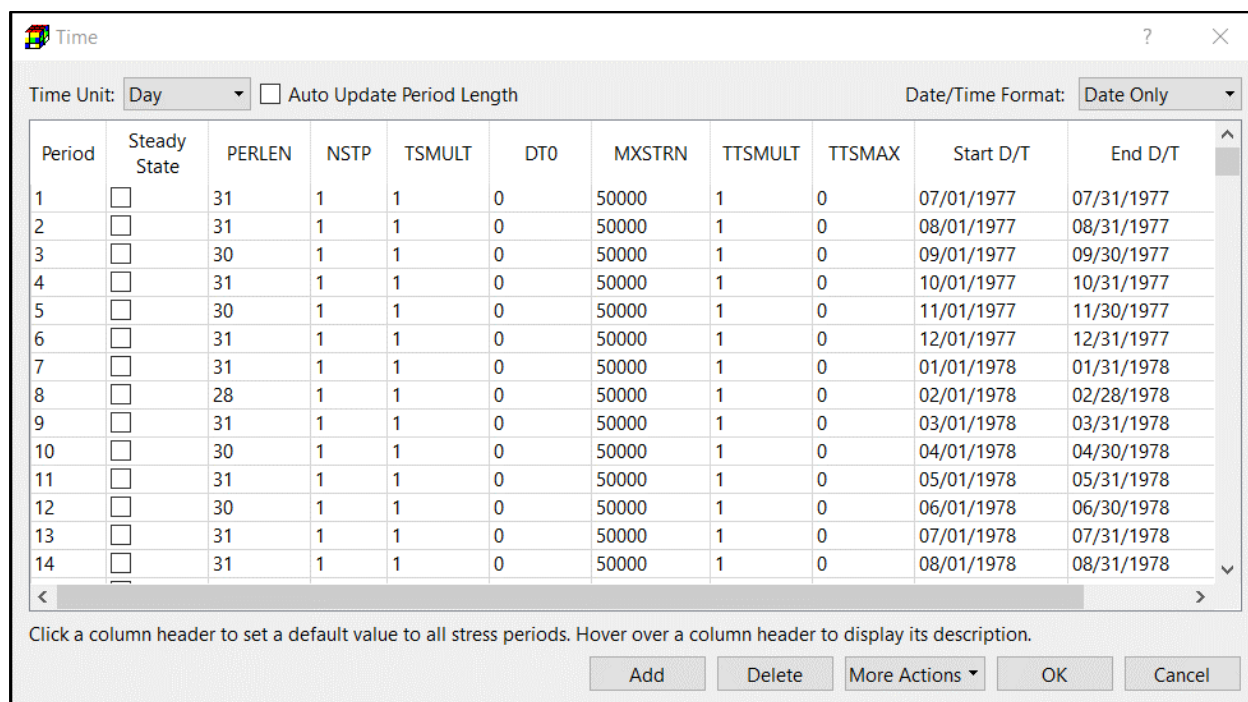


Figure 49. The Time dialog box

- **PERLEN** is the length of a stress period, expressed in the unit selected by the Time Unit dropdown box. The sum of all PERLEN values is defined as the Simulation Length of the flow simulation. In a MODFLOW simulation, Simulation Time starts at 0 and increases throughout the course of the simulation.
- **NSTP** is the number of time steps within a stress period.
- **TSMULT** is the time step multiplier, which is the ratio of the length of each time step to that of the preceding time step. The length of the first time step, Δt_1 , is related to PERLEN, NSTP, and TSMULT by the relation:

$$\Delta t_1 = PERLEN \cdot \left(\frac{TSMULT - 1}{TSMULT^{NSTP} - 1} \right) \quad (5)$$

- **DT0** is the initial transport step length within each time step of the flow solution. If DT0 is specified as zero, the actual value of DT0 is calculated based on the Courant number specified in the Advection package. The subsequent transport step size depends on the value of TTSMULT and the solution scheme for the advection term.
- **MXSTRN** is the maximum number of transport steps allowed within a time-step of the flow solution. If the number of transport steps within a flow time step exceeds MXSTRN, the simulation is terminated.
- **TTSMULT** is the multiplier for successive transport steps within a flow time-step if the solution option for the advection term is the standard finite-difference method. A value between 1.0 and 2.0 is generally adequate. For the particle-tracking-based solution options and the third order TVD scheme, TTSMULT does not apply.
- **TTSMAX** is the maximum transport step size allowed when transport step size multiplier TTSMULT > 1.0. The limit is lifted by setting TTSMAX = 0.

- **Start D/T** is the starting date/time of a stress period.
 - **End D/T** is the end date/time of a stress period.
5. **Add:** Click this button to add a row (i.e., a stress period) to the table.
 6. **Delete:** Click this button to delete the selected row from the table.
 7. **More Actions > Save:** Save the contents of the table to a *Temporal Discretization* file. See Section 211109.2 for the file format.
 8. **More Actions > Load:** Populate the table with the contents of a Temporal Discretization file that was previously created with the *More Actions > Save*.
 9. **OK:** Accept and save the changes.

3.7.2. Grid

Select this menu item to edit the model grid. See the Grid Editor (Section 3.4.1) for details.

3.7.3. Layer Property

The menu item opens the Layer Property dialog box (Figure 50). You can specify the properties of individual layers in the table. The columns of the table are as follows.

1. **Layer** is the layer number. Model layers are numbered from the top down.
2. **Layer Type** affects the numerical formulation of the groundwater flow equation. The available options and required parameters depend on the selected groundwater flow package. Note that S, S_x, and S_y are used only in transient flow simulations.
 - **BCF Package** supports the following layer types:
 - **Confined:**
 - Required input parameters: T and S

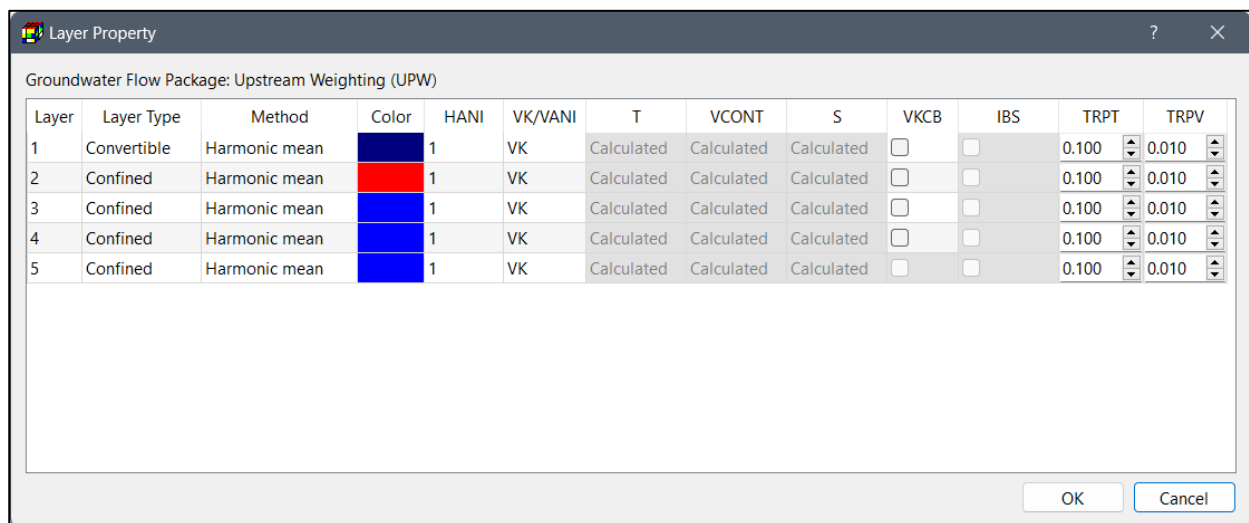


Figure 50. The Layer Property dialog box

- Interblock conductance is calculated based on input values of T, and therefore remains unchanged for the entire simulation.
- **Unconfined:** The option is valid only for layer 1.
 - Required input parameters: HK and Sy.
 - Interblock conductance is calculated based on HK and saturated thickness that may vary during a flow simulation.
- **Limited Convertible** between confined and unconfined.
 - Required input parameters: T, S, and Sy.
 - Interblock conductance is calculated based on input values of T, and therefore remains constant for the entire simulation regardless of the saturated thickness.
 - If the layer is fully saturated, the storage is calculated based on S; Otherwise Sy and saturated thickness are used.
 - Vertical flow from above is limited if the layer desaturates.
- **Full Convertible** between confined and unconfined.
 - Required input parameters: HK, S, and Sy.
 - If the layer is not fully saturated, the storage is calculated based on Sy and the saturated thickness; and the interblock conductance is calculated based on HK and the saturated thickness.
 - If the layer is fully saturated, the storage is calculated based on S; and the interblock conductance is calculated based on HK and the layer thickness.
 - Vertical flow from above is limited if the layer desaturates.
- **LPF** or **UPW Package** supports the following layer types:
 - **Confined:**
 - Required input parameters: HK, Ss, and VK or VANI.
 - Interblock conductance is calculated based on HK and layer thickness, and therefore remains constant for the entire simulation.
 - The storage is calculated based on Ss and layer thickness.
 - VK or VANI is used for calculating vertical conductance between layers.
 - **Convertible** between confined and unconfined.
 - Required input parameters: HK, Ss, Sy, and VK or VANI.
 - If the layer is not fully saturated, the storage is calculated based on Sy and the saturated thickness; and the interblock conductance is calculated based on HK and the saturated thickness.
 - If the layer is fully saturated, the storage is calculated based on Ss and the layer thickness; and the interblock conductance is calculated based on HK and the layer thickness.
 - VK or VANI is used for calculating vertical conductance between layers.

3. Method defines the method of calculating interblock transmissivity.

4. **Color** defines the color that is used to draw observation points and pathlines when they reside within the respective layer.
5. **HANI** defines the ratio of the HK along columns to HK along rows.
 - **BCF package:**
 - The absolute values of HANI are assigned to the TRPY array (i.e., an array containing a horizontal anisotropic factor for each layer) in the BCF's input file.
 - **LPF or UPW package:**
 - If $HANI > 0$, the specified HANI value is assigned to all cells in the layer.
 - If $HANI < 0$, the values specified in *Parameters > Horizontal Anisotropy* are used for the layer.
6. **VK/VANI** determines whether vertical hydraulic conductivity or vertical anisotropy is used for a layer.
 - **BCF package:**
 - This column is dimmed as input of VANI is not supported by BCF.
 - **LPF or UPW package:**
 - **VK** indicates that the values specified in *Parameters > Vertical Hydraulic Conductivity* are used for the layer.
 - **VANI** indicates that values specified in *Parameters > Vertical Anisotropy* (i.e., ratio of horizontal to vertical hydraulic conductivity) are used for the layer.
7. **T** determines how transmissivity values are defined.
 - **BCF package** requires transmissivity values for layers of the types **Confined** or **Limited Convertible**. To facilitate the data input, two options are available
 - **User-Specified:** The values specified in *Parameters > Transmissivity* are used for the layer.
 - **Calculated:** The product of the layer thickness (i.e., layer top elevation – bottom elevation) and the values specified in *Parameter > Horizontal Hydraulic Conductivity* (i.e., HK) are used for the layer.
 - **LPF or UPW package:**
 - This column is dimmed as input of T is not supported.
8. **VCONT** determines how vertical leakance values are defined.
 - **BCF package** requires the input of vertical leakance ($VCONT [T^{-1}]$) values to all layers, except for the bottom layer. This option determines whether user-specified or calculated VCONT values should be used. For the latter, VCONT is calculated based on VK and layer thickness.
 - **User-Specified:** The values specified in *Parameters > Vertical Leakance* are used in the simulation. In the Data Editor, VCONT between the layers i and $i+1$ is entered to the i^{th} layer.

- **Calculated:** VCONT is calculated with Equation (6). The calculated values are used in the simulation.

$$VCONT = \frac{2}{\frac{m_k}{(VK)_{k,i,j}} + \frac{m_{k+1}}{(VK)_{k+1,i,j}}} \quad (6)$$

Where $(VK)_{k,i,j}$ and $(VK)_{k+1,i,j}$ are the vertical hydraulic conductivity values of the layer k and layer k+1, respectively. m_k and m_{k+1} are the thickness of the layer k and k + 1, respectively.

- **LPF or UPW package:**
 - This column is deactivated as direct input of VCONT is not supported. These packages calculate VCONT internally base on Equation (6).
9. **S** determines how values of confined storage coefficient are defined in a transient flow simulation. When a layer is unconfined, the storage term is defined by specific yield [-] and the setting of S does not apply.
- **BCF package:** For a confined layer, these storage terms are given by the confined storage coefficient (=specific storage [L^{-1}] × layer thickness [L]).
 - **User-Specified:** The values specified in *Parameters > Storage Coefficient* are used in the simulation.
 - **Calculated:** The product of the layer thickness (i.e., layer top elevation – bottom elevation) and the values specified in *Parameter > Specific Storage* (i.e., Ss) are used for the layer.
 - **LPF or UPW package:**
 - This column is deactivated as LPF and UPW packages always requires the input of specific storage (Ss) values for confined layers.
10. **VKCB** indicates whether a layer has a Quasi-3D confining bed below it. Quasi-3D confining bed is supported by the LPF and UPW packages and is not supported by the BCF package. You should not check this box, if you plan to conduct particle tracking simulations with MODPATH or transport simulations with MT3D-USGS as they do not support Quasi-3D confining bed.
11. **IBS:** When the Interbed Storage package is activated, this option turns on/off the calculation of storage changes and compaction for specific layers.
12. **TRPT** is the ratio of the horizontal transverse dispersivity to the longitudinal dispersivity. TRPT is used by MT3D and SEAWAT.
13. **TRPV** is the ratio of the vertical transverse dispersivity to the longitudinal dispersivity. TRPV is used by MT3D and SEAWAT.

3.7.4. Layer Top Elevation and Layer Bottom Elevation

Select *Discretization > Layer Top Elevation* or *Discretization > Layer Bottom Elevation* to define, respectively, the top or bottom elevations of the model cells with the Data Editor.

3.7.5. IBOUND (Flow)

MODFLOW requires an IBOUND array, which contains a value for each model cell. A positive value in the IBOUND array defines a variable-head (i.e., active) cell where the hydraulic head is computed, a negative value defines a constant head cell and the value 0 defines a no-flow cell (i.e., inactive) cell. It is suggested to use 1 for variable-head cells, 0 for no-flow cells and -1 for constant head cells. Flux boundaries may be simulated by assigning appropriate fluxes with the Flow and Head Boundary or the Well package.

For constant head cells, the hydraulic head is hold at the starting value throughout the simulation. The starting hydraulic head is specified in *Parameters > Starting Hydraulic Heads*. A constant head boundary exists when an aquifer is in direct hydraulic contact with a river, a lake, or a reservoir in which the water (groundwater) level is known to be constant. It is important to know that constant head boundaries can provide inexhaustible supply or sink of water. Care must be taken when using constant head boundaries, as a groundwater system may get or lose unrealistically much water as necessary from or to such a boundary. It is recommended to avoid using this boundary condition on the upstream side of the groundwater flow direction. Consider using the General-Head Boundary or the Time-Variant Specified-Head packages, if the hydraulic head at the constant head boundary varies with time. Head-dependent boundary conditions may be simulated by means of the general head boundary, river, stream, reservoir or drain package.

3.7.6. ICBUND (Transport)

The transport models MT3D and SEAWAT require an ICBUND array, which contains a value for each model cell. A positive value in the ICBUND array defines an active concentration cell (the concentration varies with time and is calculated), a negative value defines a constant-concentration cell (the concentration is constant) and the value 0 defines an inactive concentration cell (no transport simulation takes place at such cells). It is recommended to use the value 1 for an active concentration cell, -1 for a constant-concentration cell, and 0 for an inactive concentration cell. Note that the ICBUND array applies to all species. Other types of boundary conditions are implemented by assigning concentrations to inflows or assigning a mass-loading rate to cells. For details, see Section 3.9.3.3.4.

No-flow or dry cells are automatically converted to inactive concentration cells. Variable-head cells may be treated as inactive concentration cells to minimize the area needed for transport simulation, if the solute transport is insignificant near those cells. For constant-concentration cells, the initial concentration remains the same at the cell throughout the simulation. A constant head cell may or may not be a constant-concentration cell. The initial concentration is specified in *Models > Transport (MT3D/SEAWAT) > Initial Concentration*.

3.8. Parameters Menu

3.8.1. Starting Hydraulic Head

MODFLOW requires a starting hydraulic head value (starting head [L]) for each model cell at the beginning of a flow simulation. Starting heads at constant head cells are used as prescribed head values and remain constant throughout the flow simulation.

For steady-state flow simulations, the starting heads are used as initial values for the iterative equation solvers and should be set as close as possible to the final head solution. For an unconfined or a convertible layer, the starting head of a constant head cell must be higher than the elevation of the cell bottom. Otherwise, MODFLOW will write the message *CONSTANT-HEAD CELL WENT DRY – SIMULATION ABORTED* to its run listing file and then abort the simulation.

In a transient flow simulation, the starting heads provide a reference elevation for the solution. It is a standard practice to use heads generated by a steady-state model as the initial conditions for a transient model, see Section 7.4 of (Anderson, et al., 2015) for more details.

3.8.2. Horizontal Hydraulic Conductivity (HK) and Transmissivity (T)

HK [L/T] is the horizontal hydraulic conductivity along model rows. T [L²/T] is the product of HK and layer thickness. Typical values and ranges of HK for diverse types of soils are given in many groundwater textbooks, for example Freeze, et al. (1979), Spitz, et al. (1996) and Fetter (1994).

If the BCF package is selected as the groundwater flow package, HK is required for *unconfined* and *full convertible* layers and T is required for *confined* or *limited convertible* layers. You have the option to specify T directly by selecting *Parameters > Transmissivity* or specify HK by selecting *Parameters > Horizontal Hydraulic Conductivity* and let PM calculate T by using the layer thickness and HK. See Section 3.7.3 for details.

If the LPF or UPW package is selected as the groundwater flow package, HK is required for all layer types and direct input of T is not possible.

3.8.3. Horizontal Anisotropy (HANI)

HANI [-] is the ratio of horizontal hydraulic conductivity along columns to horizontal hydraulic conductivity along rows.

If the BCF package is selected as the groundwater flow package, this menu item is deactivated as the BCF only supports the input of layer-by-layer HANI values and does not support the cell-by-cell HANI values. Use the Layer Property dialog box (Section 3.7.3) to input the layer-by-layer HANI values.

If the LPF or UPW package is selected as the groundwater flow package, you may use this menu item to enter the cell-by-cell HANI values. The entered values may or may not be used in a

simulation, depending on the settings of the HANI column in the Layer Property dialog box. See Section 3.7.3 for details.

3.8.4. Vertical Hydraulic Conductivity (VK) and Vertical Leakance (VCONT)

MODFLOW uses VK or VCONT to formulate the vertical flow terms.

The BCF package expects VCONT arrays in its input file. If the BCF package is selected as the groundwater flow package, you have the option to specify VCONT directly or specify VK and let PM calculate VCONT based on the setting of the VCONT column in the Layer Property dialog box. See the setting of VCONT in Section 3.7.3 for details.

The LPF and UPW packages expect VK or Vertical Anisotropy (VANI) arrays in their input files. These packages calculate the VCONT values internally based on Equation (6). A direct input of VCONT arrays is not possible and you must either specify VK or VANI values depending on the setting of VK/VANI in the Layer Property dialog box. See the setting of VK/VANI in Section 3.7.3 for details.

3.8.5. Vertical Anisotropy (VANI)

VANI is the ratio of HK to VK.

If the BCF package is selected as the groundwater flow package, this menu item is deactivated as the input of VANI is not supported.

If the LPF or UPW package is selected as the groundwater flow package, you have the option to enter VK or VANI depending on the setting of VK/VANI in the Layer Property dialog box. See the setting of VK/VANI in Section 3.7.3 for details.

3.8.6. Specific Storage (Ss) and Storage Coefficient (S)

For a steady state simulation, these menu items are not used and are deactivated. S is required by MODFLOW for confined or convertible layers.

Specific storage [L^{-1}] or *specific storativity* is defined as the volume fraction of water that a unit column of aquifer releases from storage under a unit decline in hydraulic head. The specific storage is a function of the compressibility of the water and the elastic property of the soil matrix and ranges in value from 3.3×10^{-6} [m^{-1}] of rock to 2.0×10^{-2} [m^{-1}] of plastic clay (Domenico, 1972).

In a confined layer, the storage term is given by the *storativity* or *storage coefficient* [-] (= specific storage [L^{-1}] \times layer thickness [L]). The storativity is sometimes referred to as *confined storage coefficient*.

If the BCF package is selected as the groundwater flow package, you have the option to enter S directly by selecting *Parameters > Storage Coefficient*, or enter Ss by selecting *Parameters > Specific Storage* and let PM calculate S. See the setting of S in Section 3.7.3 for details.

If the LPF or UPW package is selected as the groundwater flow package, Ss is required. A direct input of S is not possible.

3.8.7. Specific Yield (Sy)

For a steady state simulation, this menu item is not used and is deactivated. Sy is required by MODFLOW for unconfined or convertible layers.

Specific yield [-] is defined as the volume of water that an unconfined aquifer releases from storage per unit surface area of aquifer per unit decline in the water table. *Specific yield* is a function of porosity (and is not necessarily equal to porosity), because a certain amount of water is held in the soil matrix and cannot be removed by gravity drainage. Refer to Spitz, et al. (1996) for a summary of values of specific yield. Refer to Bear (1972; 1979), or Freeze (1979) for detailed explanation of storage terms and their definitions.

3.8.8. Effective Porosity (ne)

Effective porosity is used by MODPATH, MT3D and SEAWAT for calculating the average velocity of the flow through the porous medium.

If the total unit volume V of a soil matrix is divided into the volume of the solid portion V_s and the volume of voids V_v , the total porosity n is defined as $n = V_v/V$. Effective porosity (with the respect to flow through the medium) is always less than total porosity, sometimes (as in the case of clays) much less, because some of the pore spaces may be too small or too poorly connected so that part of the fluid is immobile or partially immobile. A summary of representative porosity values for different soil types can be found in Zheng, et al. (1995) or Domenico, et al. (1997).

3.8.9. Stop Zone

Stop Zone is used by MODPATH to define stopping locations for particles. The Stop Zone option of MODPATH is activated by specifying a positive Stop Zone value to MODPATH's settings window (Figure 173). Particles stop when they enter cells with the specified Stop Zone value.

3.8.10. Retardation Factor (R)

Retardation factor is used by MODPATH to alter flow velocity values at model cells. The altered flow velocity values are used in particle tracking simulation. A retardation factor greater than 1 slows down the simulated particles. A retardation factor less than 1 speeds up the simulated particles.

Retardation factor is a function of porosity, density of the geologic material, and soil sorption coefficient. A detailed discussion of retardation factor can be found in (Domenico, et al., 1997). For nonreactive solutes that do not interact with the surrounding rock, the retardation factor is usually equal to 1 and the magnitude of the apparent solute velocity is equal to that of the groundwater velocity. For solutes affected by sorption, the retardation factor is greater than 1 and the magnitude of the apparent solute velocity is less than that of the groundwater velocity.

3.8.11. Bulk Density

The cell-by-cell bulk density data are used by the Reaction package of MT3D and SEAWAT for simulating sorption effects.

3.8.12. Static Sandboxes

A Static Sandbox is a place where you can store data for visualization purposes. The values specified in a Static Sandbox do not vary with time and will not be used in simulation.

3.8.13. Transient Sandboxes

A Transient Sandbox is a place where you can store time-dependent data for animation and visualization purposes. For a model with multiple stress periods, the user may assign various matrix values to (the end of the selected) stress periods. If a contour map is created with the data of transient sandbox, the contour map will transit between stress periods based on the linearly interpolated matrix values of those stress periods.

3.9. Models Menu

3.9.1. MODFLOW

3.9.1.1. Packages

A MODFLOW package is a component of the program that handles a specific aspect of the simulation. For instance, the Well Package simulates the influence of wells on the groundwater system, while the River Package simulates the influence of rivers. To activate and edit the data associated with a package, you can select the respective menu item from the *Models > Modflow > Packages* menu. If the package is already activated, you will be given the option to either edit its data or deactivate the package. Additionally, you can right-click on a package in the TOC to display a context menu with the following options: Activate Package, Deactivate Package, Remove Package, Export to Shapefile⁴³. These options give you flexibility in managing the simulation packages and exporting data when necessary.

3.9.1.1.1. Drain (DRN)

The Drain (DRN) Package is used to simulate effects of features such as agricultural drains, which remove groundwater from aquifer at a rate proportional to the head difference between the aquifer and the drain.

A drain is implemented by dividing into drain segments so that each drain segment is completely contained in a single model cell. Multiple drain segments may be assigned to a model cell. For

⁴³ This option is available only for these packages: DRN, DRT1, FHB1, GHB, HFB, MNW1, MNW2, RIV, CHD, WEL, RES, SFR2, STR, and LAK3.

each drain segment, the recharge rate from the drain to the model cell is always zero, and the discharge rate from the model cell to the drain is calculated by

$$Q_d = C_d \cdot (h - h_d) \quad \text{if } h > h_d \quad (7)$$

$$Q_d = 0 \quad \text{if } h \leq h_d \quad (8)$$

where

Q_d [L^3T^{-1}] is the discharge rate from the cell to the drain segment,

h [L] is the calculated hydraulic head in the cell containing the drain segment,

h_d [L] is the elevation of the drain segment, and

C_d [L^2T^{-1}] is the conductance of the drain segment.

The required input parameters are listed below.

1. Cell-by-cell input method: Figure 51 shows the dialog box for specifying one or more drain segments to a model cell. For each segment, the following properties need to be assigned.

- **Drain conductance (C_d) [L^2T^{-1}]** relates drain discharge rate to the head loss between the drain and the aquifer. The value of C_d depends on the material and characteristics of the drain and the immediate environment.
- **Drain elevation (h_d) [L]** is the elevation of the drain segment.
- **Parameter Number (PEST) [-]** is used to group cells, where the C_d value is to be estimated by PEST.
- **Elevation of drain bottom [L]** is used by SEAWAT to calculate reference head considering the density effect to accurately simulate the flow of variable-density ground water to a drain. For details, see DRNBELEV on the VDF tab of the Transport Settings dialog box (Figure 119).

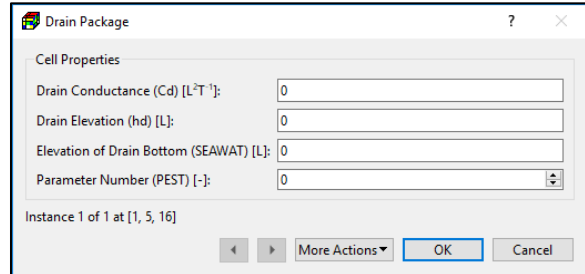


Figure 51. The Drain Package dialog box for specifying cell properties

2. **Polyline input method:** Figure 52 shows the dialog box that is used to specify polyline and vertex properties.

- **Polyline Properties**

- **Auto Layer Number:**

- If this box is checked, the layer number of a drain segment along the polyline is determined by comparing its drain elevation (h_d) with the top/bottom elevations of the cells in the same vertical model column as that drain segment.

- if h_d is higher than the top elevation of the highest cell, the drain segment is assigned to the cell in layer 1.
 - if h_d is lower than the bottom elevation of the lowest cell, the drain segment is assigned to the cell in layer NLAY.
 - If h_d lies between top and bottom elevations of a cell, the drain segment is assigned to that cell.

- If this box is not checked, the drain segments along the polyline are assigned to the layer where the polyline is located.

- **Vertex Properties:** The specified vertex data may be applied to model cells following the rules described in Section 3.4.2.2 and the rules below.

- **Drain Conductance per Unit Length [LT⁻¹].** When applying the vertex data to model cells, the specified value is interpolated to the DRN cells along the polyline. For each of those cells, the interpolated cell value is multiplied with the length of the drain within the cell to obtain the drain conductance C_d .

$$C_d = C_{dl} \cdot L_c \tag{9}$$

where

C_{dl} is the drain conductance per unit length at a given cell, and

L_c is the length of the drain within the cell.

- **All other parameters:** Refer to the cell-by-cell input method above for the descriptions of these parameters.

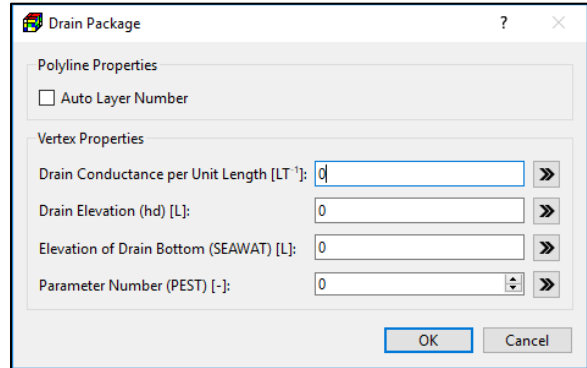


Figure 52. The Drain Package dialog box for specifying vertex properties

3. **Polygon input method:** Figure 53 shows the dialog box that is used to specify polygon properties. The polygon data may be assigned to model cells following the rules described in Section 3.4.2.3 and the rules below.

- **Auto Layer Number:** See the description for the polyline input method above.
- **Drain Conductance per Unit Area [T^{-1}]:** When applying the polygon data to model cells within the polygon, the specified value (C_{da}) is multiplied with the cell area to obtain the drain conductance C_d .

$$C_d = C_{da} \cdot Area_c \quad (10)$$

where

C_{da} is the Drain Conductance per Unit Area,

$Area_c$ is the area of the cell.

- **All other parameters:** Refer to the cell-by-cell input method above for the descriptions of these parameters.

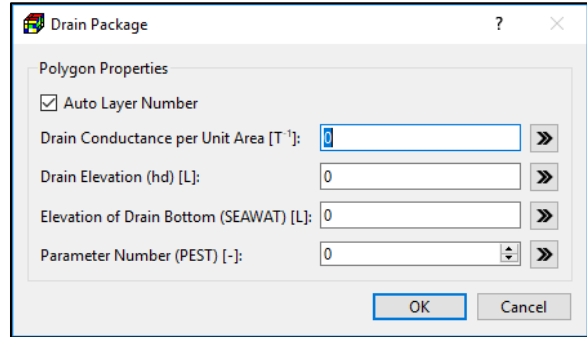


Figure 53. The Drain Package dialog box for specifying polygon properties

3.9.1.1.2. Drain with Return Flow (DRT1)⁴⁴

The Drain with Return Flow (DRT1) Package (Banta, 2000) can be used to simulate the return flow of water discharged from a drain back into the ground-water system. If the return-flow option of the DRT1 Package is enabled, for each cell designated as a drain-return cell, the user has the option of specifying a proportion of the water, simulated as leaving the ground-water system through the drain feature, to return to one other cell in the model.

The numerical formulation and most required parameters for the DRT1 Package are the same as those of the DRN Package. The parameters for the return flow function described below.

1. **All input methods:** The data entry dialog boxes of all input methods include a Simulation Settings group that contains the following options:
 - Activate the Return-flow Option of the Drain Return Package: Check this box to enable the return-flow feature of DRT.
 - **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML^{-3}]** are associated

⁴⁴ The DRT1 and MNW1 packages may not be used as the same time with MT3DMS due to incompatibility between the LINK-MT3DMS (LMT) Package of MODFLOW and the Flow Model Interface (FMI) Package of MT3DMS.

with the return flow. If this box is not checked, the calculated concentration at the outflow cell will be associated with the return flow.

2. **Cell-by-cell input method:** Figure 54 shows the dialog box for specifying cell properties.

- **Return Flow**

- If the *Activate the Return-flow for the current Instance* box is checked, the current drain instance will be able to return a portion of water to the model cell at the [Layer, Row, Column]
- **[Layer, Row, and Column]** defines the recipient cell where a portion of drain flow is returned to the groundwater system.

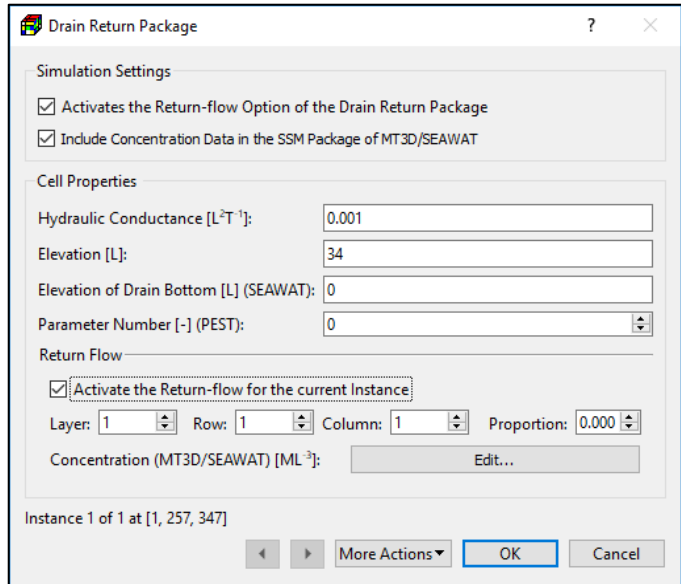


Figure 54. the Drain Return Package dialog box for specifying cell properties

- **Proportion:** The return-flow is equal to product of this value and the simulated drain flow.

- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the **Edit** button to open the Concentration dialog box (Figure 55) that allows you to specify the return flow concentration of up to 100 species. You can either enter individual concentration values to the table directly or click the *Set Value* button to enter a uniform value for all species. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

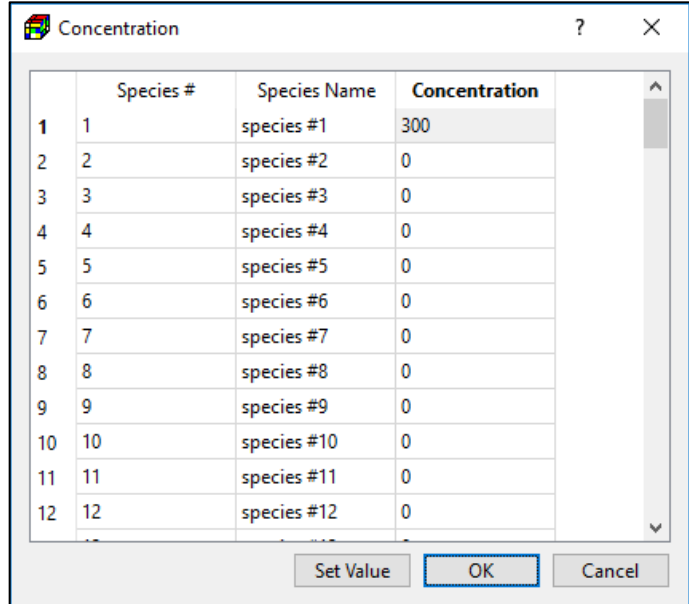


Figure 55. The Concentration dialog box

3. **Polyline input method:** Figure 56 shows the dialog box that is used to specify vertex/polyline properties.

- **Polyline Properties:** If the *Activate the Return-Flow Option for the current Polyline* box is checked, the drain instances along the polyline will be able to return a portion of water to the model cells defined by [Layer, Row, Column] in the Return Flow below.
- **Vertex Properties:**
 - **Return Flow**
 - [Layer, Row, and Column]: defines the recipient cell(s) of return-flow.
 - **Proportion:** The return-flow is equal to product of this value and the simulated drain flow.
 - **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the **Edit** button to open a Concentration dialog box (Figure 55) that allows you to specify the return flow concentration of up to 100 species to the current vertex.

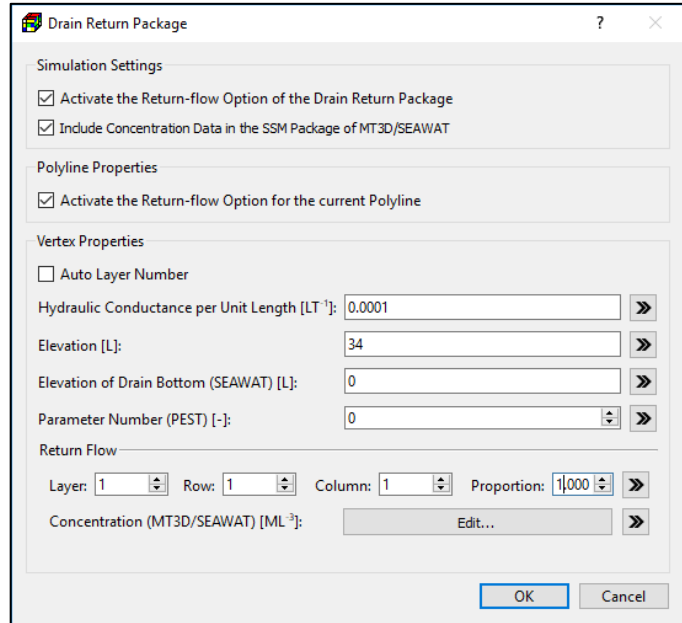


Figure 56. The Drain Return Package dialog box for specifying vertex/polyline properties

4. **Polygon input method:** Figure 57 shows the dialog box that is used to specify polygon properties.

- **Simulation Settings:** Check the *Activate the Return-flow Option of the Drain Return Package* box to enable the return-flow feature of DRT.
- **Polygon Properties:** If the *Activate the Return-Flow Option for the current Polygon* box is checked, the drain instances within the polygon will be able to return a portion of water to the model cells defined by [Layer, Row, Column] in the Return Flow below.

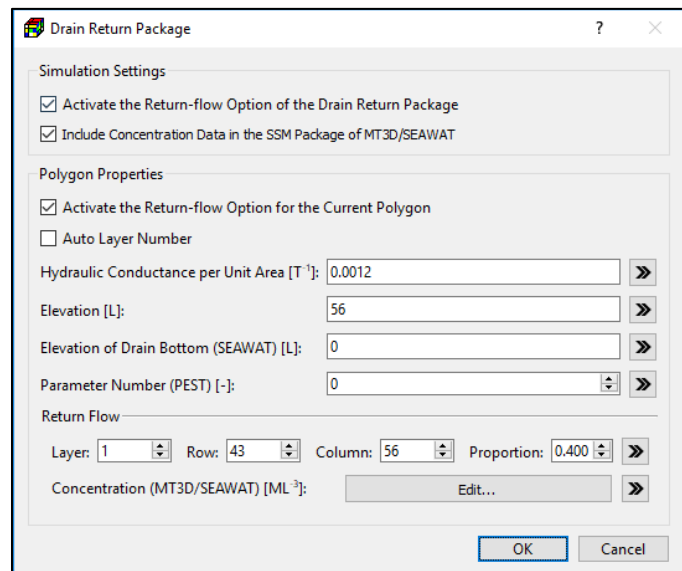


Figure 57. The Drain Return Package dialog box for specifying polygon properties

- **Return Flow**

- **[Layer, Row, and Column]**: defines the recipient cell of return-flow.
- **Proportion**: The return-flow is equal to product of this value and the simulated drain flow.
- **Concentration (MT3D/SEAWAT) [ML⁻³]**: Click the **Edit** button to open a Concentration dialog box (Figure 55) that allows you to specify the return flow concentration of up to 100 species to the current polygon.

3.9.1.1.3. Evapotranspiration (EVT)

The Evapotranspiration (EVT) Package simulates the effects of plant transpiration and direct evaporation by removing water from the saturated groundwater regime. Evapotranspiration can be drawn from only one cell in each vertical column of model cells. The evapotranspiration rate is calculated for a cell in each vertical column of model cells by

$$R_{et} = EVTR \quad \text{if } h > SURF \quad (11)$$

$$R_{et} = EVTR \cdot \frac{h - (SURF - EXDP)}{EXDP} \quad \text{if } (SURF - EXDP) \leq h \leq SURF \quad (12)$$

$$R_{et} = 0 \quad \text{if } h < SURF - EXDP \quad (13)$$

where

R_{et} [LT⁻¹] is the rate of loss per unit surface area of water table due to evapotranspiration;

h [L] is the calculated hydraulic head in the cell from which the evapotranspiration occurs;

$EVTR$ [LT⁻¹] is the maximum possible value of R_{et} or potential evapotranspiration rate;

$SURF$ [L] is the ET surface elevation;

$EXDP$ [L] is the cutoff or extinction depth.

The evapotranspiration flow rate Q_{et} [L³T⁻¹] drawn from the model cell is

$$Q_{et} = R_{et} \times Area_c \quad (14)$$

where $Area_c$ is the area of the model cell.

The required input parameters for all input methods are identical and listed below. Although these parameters are to be assigned to each vertical column of model cells, you may move between layers within the Data Editor to examine the grid configuration in each layer.

1. **All input methods:** Figure 58 shows the dialog box for specifying cell, vertex, or polygon properties.

- **Data Tab**

- **ET Option (NEVTOP) [-]:** Since Q_{et} is calculated for only one cell in each vertical column of model cells, three options are available for specifying the cell where Q_{et} is drawn from:

- ET is calculated only for cells in the top grid layer.
- ET is calculated for cells in the layers defined by IEVT (see below)
- ET is calculated for the highest variable-head cell in each vertical column.

- **Maximum ET Flux (EVTR) [LT^{-1}]:** potential evapotranspiration rate.
- **Elevation of ET Surface (SURF) [L]:** when head is higher than SURF, R_{et} equals to EVTR.
- **ET Extinction Depth (EXDP) [L]:** when head is lower than SURF - EXDP, R_{et} is zero.
- **Layer Indicator (IEVT) [-]:** This indicates the layer from which Q_{et} is removed from the groundwater system.
- **Parameter Number (PEST)** is used to group cells, where the EVTR value is to be estimated by PEST.

- **Options Tab:** This tab contains the following options. You can check the options to use the respective data of the first stress period for the entire simulation even if the data of other stress periods have been specified.

- Use Maximum ET Flux (EVTR) of the first stress period for the entire simulation
- Use Elevation of ET Surface (SURF) of the first stress period for the entire simulation
- Use ET Extinction Depth (EXDP) of the first stress period for the entire simulation
- Use Layer Indicator (IEVT) of the first stress period for the entire simulation

3.9.1.1.4. Evapotranspiration Segments (ETS1)

The Evapotranspiration Segments (ETS1) Package (Banta, 2000) allows simulation of evapotranspiration with a user-defined relation between evapotranspiration rate and hydraulic head. In the ETS1 Package, the relation of evapotranspiration rate to hydraulic head is conceptualized as a segmented line between an evaporation surface, defined as the elevation where the evapotranspiration rate reaches a maximum, and an elevation located at an extinction depth below the evaporation surface, where the evapotranspiration rate reaches zero. The user supplies input to define as many intermediate segment endpoints as desired to define the

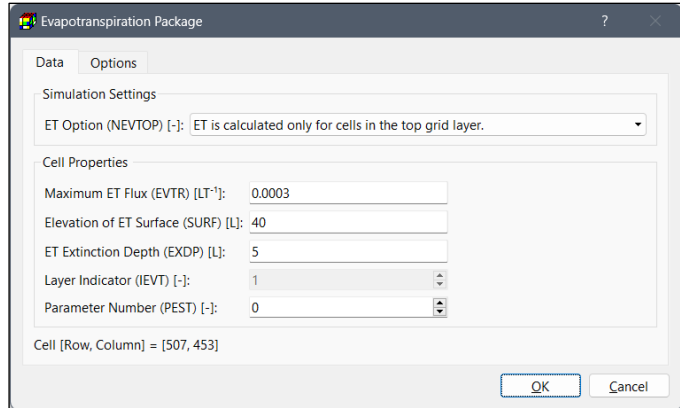


Figure 58. The Evapotranspiration Package dialog box

relation of evapotranspiration rate to head between these two elevations. The EVT Package, in contrast, simulates evapotranspiration with a single linear function.

The numerical formulation and most required parameters for the ETS Package are the same as those for the EVT Package, other than the segmented function. The required input parameters for all input methods identical and are listed below. Although these parameters are to be assigned to each vertical column of model cells, you may move between layers within the Data Editor to examine the grid configuration in each layer.

1. All input methods: Figure 59 shows the dialog box for specifying cell, vertex, or polygon properties.

- **Data Tab**

- **ET Option (NETSOP) [-]:** Since Q_{et} is calculated for only one cell in each vertical column of model cells, three options are available for specifying the cell where Q_{et} is drawn from:
 - ET is calculated only for cells in the top grid layer.
 - ET is calculated for cells in the layers defined by IETS (see below)
 - ET is calculated for the highest variable-head cell in each vertical column.
- **Number of Segments [-]:** This is the number of segments of a segmented function that defines the relation of ET rate to hydraulic head. The segmented function is defined by the PXDP and PETM values in the **ET Segment Properties** table below.
- **Maximum ET Flux (ETSR) [LT^{-1}], Elevation of ET Surface (ETSS) [L], and ET Extinction Depth (ETSX) [L]:**
 - When the calculated head is higher than ETSS, the ET rate equals to ETSR.
 - When the calculated head is lower than ETSS - EXSS, the ET rate is zero.
 - When the calculated head is below ETSS but the depth of the head below ETSS is less than ETSX, the ET rate is variable and depends on the segmented function defined by the **ET Segment Properties** table below.
- **Layer Indicator (IETS) [-]:** This indicates the layer from which Q_{et} is removed

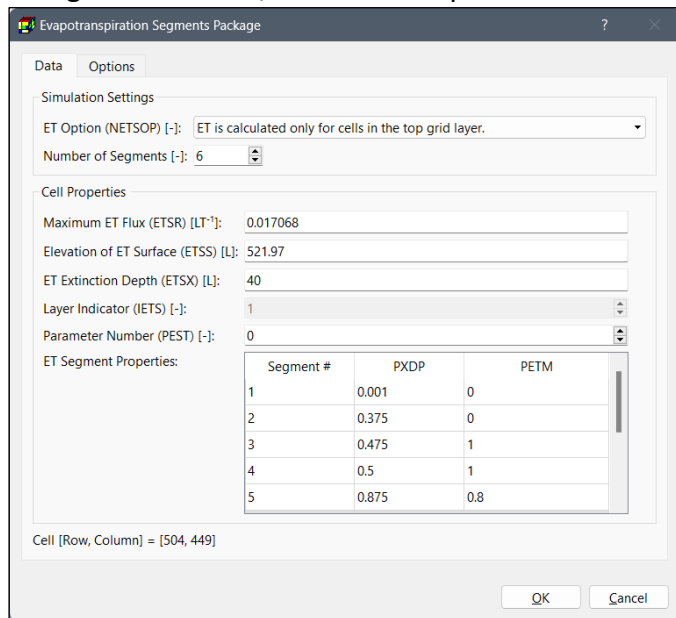


Figure 59. The Evapotranspiration Segments Package dialog box

from the groundwater system.

- **Parameter Number (PEST)** is used to group cells, where the ETSR value is to be estimated by PEST.
- **ET Segment Properties:**
 - **PXDP** is a proportion of the extinction depth (dimensionless), measured downward from the ET surface, which, with PETM, defines the shape of the relation between the evapotranspiration rate and head. PXDP is 0.0 at the ET surface and is 1.0 at the extinction depth. Repetitions of PXDP and PETM are to be specified in sequence such that the first occurrence represents the bottom of the first segment, and subsequent repetitions represent the bottom of successively lower segments. Accordingly, PXDP values for later repetitions (representing lower segments) should be greater than PXDP values for earlier repetitions.
 - **PETM** is a proportion of the maximum evapotranspiration rate (dimensionless) which, with PXDP, defines the shape of the relation between the evapotranspiration rate and head. PETM is 1.0 at the ET surface and is 0.0 at the extinction depth. Repetitions of PXDP and PETM are to be specified in sequence such that the first occurrence represents the bottom of the first segment, and subsequent repetitions represent the bottoms of successively lower segments. Accordingly, PETM values for later repetitions (representing lower segments) generally would be less than PETM values for earlier repetitions.
- **Options Tab:** This tab contains the following options. You can check the options to use the respective data of the first stress period for the entire simulation even if the data of other stress periods have been specified.
 - Use Maximum ET Flux (ETSR) of the first stress period for the entire simulation
 - Use Elevation of ET Surface (ETSS) of the first stress period for the entire simulation
 - Use ET Extinction Depth (ETSX) of the first stress period for the entire simulation
 - Use Layer Indicator (IETS) of the first stress period for the entire simulation
 - Use ET Segment Properties of the first stress period for the entire simulation

3.9.1.1.5. Flow and Head Boundary (FHB1)

The Flow and Head Boundary (FHB1) Package (Leake, et al., 1997) allows you to specify flow or head boundary conditions that vary at times other than starting and ending times of stress periods and associated time steps. Values of flow and (or) head at each time step are calculated by linear interpolation of user-specified time-series values. The ability to assign variable flow and head conditions defined at times not corresponding with the model stress periods allows greater flexibility in simulating natural geohydrologic systems and, at the same time, improves the efficiency of the methods used to represent these systems.

The FHB1 Package requires the input of an array of simulation times and for each of the simulation time, the predefined head or flow values at an instance of the FHB1 feature. Multiple instances of FHB1 features may be assigned to a model cell.

The FHB1 Package can be used as an alternative and/or supplement to the Well (WEL) Package or the Recharge (RCH) Package for simulating specified-flow boundaries. FHB1 also is an alternative and/or supplement to the Time-Variant Specified-Head (CHD) Package for simulating specified-head boundaries.

The required input parameters are listed below.

1. All input methods: The data entry dialog boxes (Figure 61, Figure 63, Figure 64) of all input methods include a Simulation Settings group that contains the following options:

- **Times of Specified Values (BDTIM):** Click the Edit button to open the BDTIM dialog box (Figure 60) where you can edit an array of simulation times at which the flow or head values can be defined. The array of simulation times applies to the entire package. The features of the BDTIM dialog box are as follows:

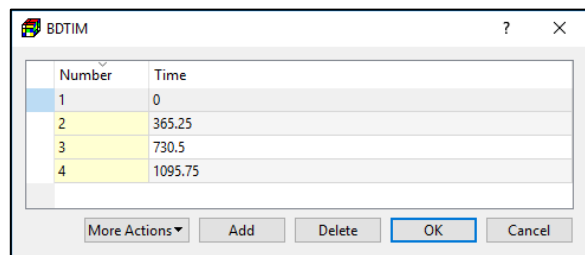


Figure 60. The BDTIM dialog box

- **Table** contains an array of simulation times.
 - **Add:** Click to add a row to the table.
 - **Delete:** Click to delete the selected row(s) from the table.
 - **More Actions > Save:** Select this item to save the defined simulation times in a Series file (see Section 5.1.14 for its format).
 - **More Actions > Load:** Select this item to clear the table and load simulation times from a Series file.
- **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML⁻³]** are associated with the source water. If this box is not checked, the concentration of the source water is assumed to be zero.

2. **Cell-by-cell input method:** Figure 61 shows the dialog box for specifying one or more FHB1 instances to a model cell.

- **Boundary Type** defines whether the cell represents a specified flow or a specified head⁴⁵ boundary, or none. If the UZF1 package is used, its IUZFBND should be set to zero for all specified head cells.
- **Cell Face (IAUX)** can be used by programs such as MODPATH (Pollock, 2016) to store information such as the cell face associated with the specified-flow or specified-head boundary.
- **Parameter Number (PEST) [-]** is used to group cells, where a multiplier for specified flows or an addend to specified heads will be estimated by PEST. The default value for a multiplier is 1; the default addend is 0. During a PEST run, the product of the estimated multiplier and the specified-flow values as well as the sum of the estimated addend and the specified-head values are used by the model.
- **Time Series of Specified-Flow or Time Series of Specified-Head:** The Edit button is activated only if the Boundary Type is Specified-Head or Specified-Flow. Click the Edit button to open the Time Series dialog box (Figure 62) where you can edit an array of specified-flow or specified-head values for the current FHB1 cell instance. The features of the Time Series dialog box are as follows:

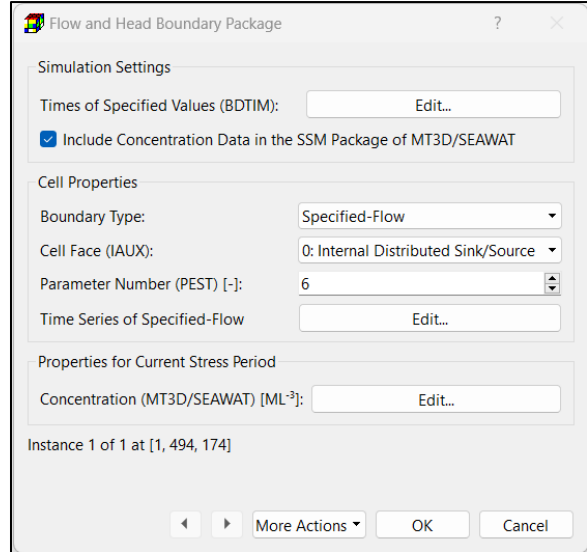


Figure 61. The Flow and Head Boundary Package dialog box for specifying cell properties

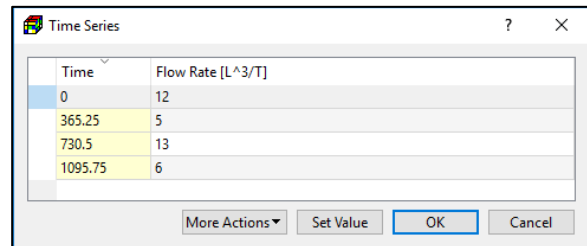


Figure 62. The Time Series dialog box

- **Table** contains an array of (read-only) times defined in the BDTIM dialog box and the corresponding flow or head values for the current FHB1 cell instance.
- **Set Value:** Click to set a uniform head or flow value to the selected row(s) of the table.
- **More Actions > Save:** Select this item to save the defined flow or head values in a Series file (see Section 5.1.14 for its format).
- **More Actions > Load:** Select this item to load the flow or head values from a Series file.

⁴⁵ If the UZF package is used, IUZFBND should be set to zero for all specified head cells.

- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species to the current FHB1 instance. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

3. **Polyline input method:** Figure 63 shows the dialog box that is used to specify vertex/polyline properties.

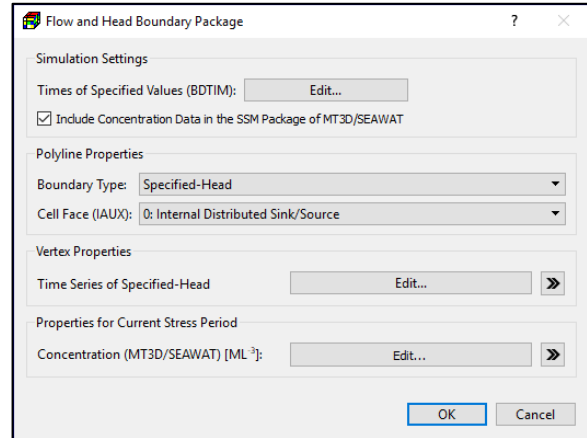


Figure 63. The Flow and Head Boundary Package dialog box for specifying vertex

- **Polyline Properties**
 - **Boundary Type** defines whether the FHB1 instances in the cells along the polyline represent specified flow or specified head⁴⁵ boundary conditions, or none.
 - **Cell Face (IAUX)** will be assigned to FHB1 instances in the cells along the polyline. IAUX can be used by programs such as MODPATH (Pollock, 2016) to store information such as the cell face associated with the specified-flow or specified-head boundary.
- **Vertex Properties:** The specified vertex data may be applied to model cells following the rules described in Section 3.4.2.2.
 - **Time Series of Specified-Flow** or **Time Series of Specified-Head:** Click the Edit button to edit an array of specified flow or head values at the current vertex for each of the times defined by BDTIM.
 - **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species to the current vertex. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

4. **Polygon input method:** Figure 64 shows the dialog box that is used to specify polygon properties. These data may be applied to model cells following the rules described in Section 3.4.2.3.

- **Polygon Properties**

- **Boundary Type** defines whether the cells within the polygon represent specified flow or specified head boundary conditions, or none.
- **Cell Face (IAUX)** will be assigned to cells within the polygon. IAUX can be used by programs such as MODPATH (Pollock, 2016) to store information such as the cell face associated with the specified-flow or specified-head boundary.
- **Time Series:** Click the Edit button to edit an array of specified flow or head values at the current vertex for each of the times defined by BDTIM. The specified polygon data are assigned to model cells within the polygon following the rules described in Section 3.4.2.3.
- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species to the current polygon. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

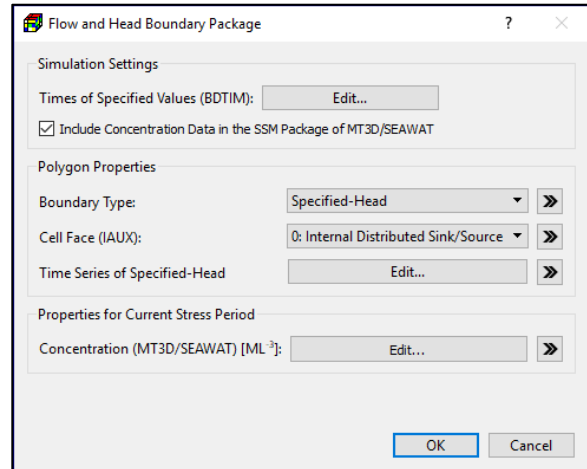


Figure 64. The Flow and Head Boundary Package for specifying polygon properties

3.9.1.1.6. General-Head Boundary (GHB)

The General-Head Boundary (GHB) Package is used to simulate head-dependent flow boundaries (Cauchy boundary conditions), where flow between an external boundary and the aquifer is at a rate proportional to the head difference between the external boundary and the aquifer.

An external boundary is implemented by dividing into segments so that each segment is completely contained in a single model cell. Multiple external boundary segments may be assigned to a model cell. For each segment, the flow from the external boundary to the cell is calculated by

$$QB = CB \cdot (HB - h) \quad (15)$$

where

- QB [L³T⁻¹] is the flow into the cell from the segment of external boundary,
- CB [L²T⁻¹] is the boundary conductance of the segment of external boundary,
- HB [L] is the head at the segment of external boundary, and

h [L] is the calculated hydraulic head in the cell.

The required input parameters are listed below.

1. **All input methods:** The data entry dialog boxes (Figure 65, Figure 66, and Figure 67) of all input methods include a Simulation Settings group that contains the following option.

- **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML⁻³]** are used as source water concentration. If this box is not checked, the concentration of the source water is assumed to be zero.

2. **Cell-by-cell input method:** Figure 65 shows the dialog box for specifying one or more external boundary segments to a model cell. For each segment, the following properties need to be assigned.

- **Boundary conductance (CB) [L²T⁻¹].** The value of CB depends on the characteristics of the aquifer and the external boundary.
- **Hydraulic Head on Boundary (HB) [L].** This is the head at the segment of external boundary.
- **Parameter Number (PEST)** is used to group cells, where the CB value is to be estimated by PEST.
- **GHB Elevation (SEAWAT) [L]:** This is the elevation of the boundary from which the equivalent reference head is calculated. This value is used by SEAWAT to accurately calculate the flow of variable-density groundwater to the external boundary. The specified value is used only if GHBELEV (in Figure 119) is set as *User-Specified in the GHB Package*.
- **Water Density at GHB (SEAWAT) [L]:** This value is used by SEAWAT and represents the prescribed density of fluid entering the groundwater system from the general-head boundary. This value is used only if SEAWAT is running in the uncoupled mode (i.e., the density-effect of all species are turned off, see Section 3.9.3.1) and GHBDENS (in Figure 119) is set as *User-Specified in the GHB Package*.
- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

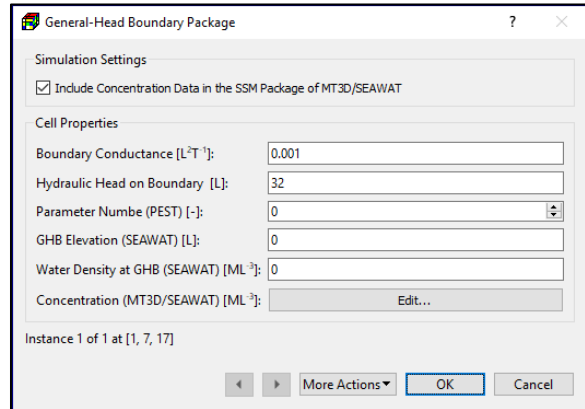


Figure 65. The General-Head Boundary Package dialog box for specifying cell properties

3. **Polyline input method:** Figure 66 shows the dialog box for specify data to one or more vertices of a polyline.

- Simulation Settings

- Auto Layer Number

- If this box is checked, the layer number of a segment of external boundary along the polyline is determined by comparing its *hydraulic head on external boundary (HB)* with the top/bottom elevations of the cells in the same vertical model column as that segment.

- if HB is higher than the top elevation of the highest cell, the segment is assigned to the cell in layer 1.
 - if HB is lower than the bottom elevation of the lowest cell, the segment is assigned to the cell in layer NLAY.
 - If HB lies between top and bottom elevations of a cell, the segment is assigned to that cell.

- If this box is not checked, the external boundary segments along the polyline are assigned to the layer where the polyline is located.

- **Vertex Properties:** The data of this group are assigned to one or more vertices of a polyline. These vertex data may be interpolated to model cells following the rules described in Section 3.4.2.2 and the rules below.

- **Boundary conductance per unit length [LT⁻¹]** is interpolated to the cells along the polyline. For each of those cells, the interpolated cell value CB_l is multiplied with the length of the boundary within the cell to obtain the boundary conductance C_d .

$$CB = CB_l \cdot L_c \tag{16}$$

where

CB_l is the boundary conductance per unit length at a given cell, and

L_c is the length of the boundary within the cell.

- **All other parameters:** Refer to the cell-by-cell input method above for the descriptions of these parameters.

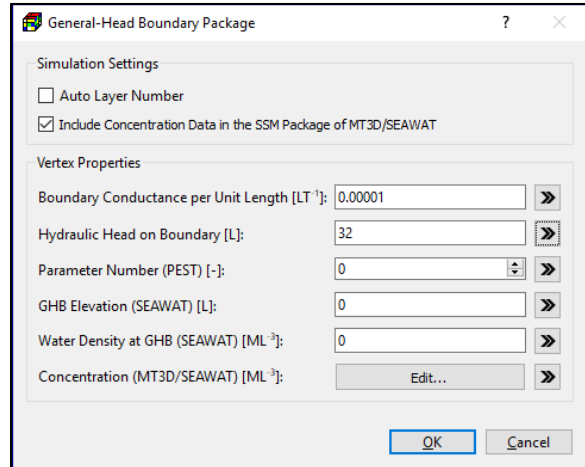


Figure 66. The General-Head Boundary Package dialog for specifying vertex properties.

4. **Polygon input method:** Figure 67 shows a dialog box for specify the following data to a polygon. These data may be applied to model cells following the rules described in Section 3.4.2.3 and additional rules below.

- **Simulation Settings:** See the description for the polyline input method above.
- **Polygon Properties:**

- **Boundary conductance per unit area [T^{-1}].** For each cell within the polygon, the specified value (CB_a) is multiplied with the cell area to obtain the boundary conductance CB .

$$CB = CB_a \cdot Area_c \tag{17}$$

where

CB_a is the boundary conductance per unit area [T^{-1}], and $Area_c$ is the area of the cell.

- **The other parameter:** Refer to the cell-by-cell input method above for the descriptions of these parameters.

3.9.1.1.7. Horizontal-Flow Barrier (HFB6)

The Horizontal-Flow Barrier (HFB6) Package simulates thin low-permeability geologic features, such as vertical faults or slurry walls, which impede the horizontal flow of groundwater. These geologic features are approximated as a series of horizontal-flow barriers conceptually situated on the boundaries between pairs of adjacent cells in the finite-difference grid. The Horizontal-Flow Barrier was developed by (Hsieh, et al., 1993) for an earlier version of MODFLOW. The package was renamed to HFB6 and incorporated into MODFLOW–2005 and MODFLOW-NWT.

The required input parameters are listed below.

1. **Cell-by-cell input method:** Figure 68 shows the dialog box for specifying cell properties.

- **Hydraulic characteristic [$1/T$]** is the hydraulic conductivity of the

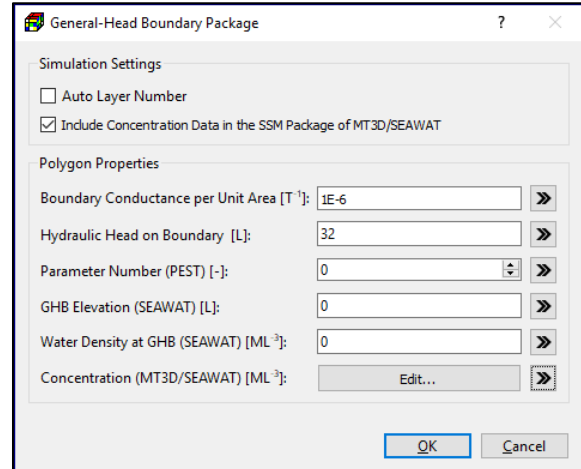


Figure 67. The General-Head Boundary Package dialog for specifying polygon properties.

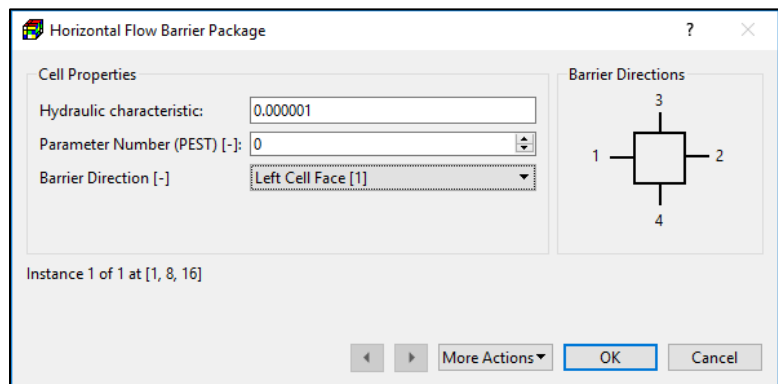


Figure 68. The Horizontal Flow Barrier Package dialog box

barrier divided by the thickness of the barrier.

- **Parameter Number (PEST)** is used to group cells, where the hydraulic characteristic value is to be estimated by PEST.
- **Barrier Direction [-]:** The barrier direction indicates the cell face where the barrier is located. To erase an existing barrier, select No Barrier [0].

2. Polyline input method:

Figure 69. The Horizontal Flow Barrier Package dialog box for specifying vertex properties. Figure 69 shows the dialog box for specifying vertex properties, which may be applied to model cells following the rules described in Section 3.4.2.2. See Cell-by-cell input method above for the definition of the properties.

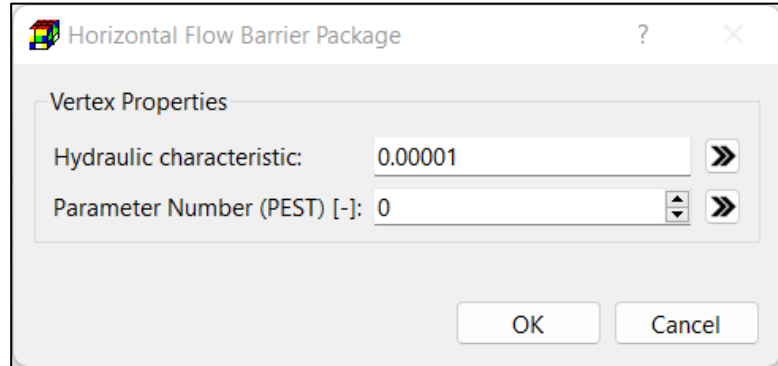


Figure 69. The Horizontal Flow Barrier Package dialog box for specifying vertex properties

3. Polygon input method: This method is not supported.

3.9.1.1.8. Interbed Storage (IBS)

The Interbed Storage (IBS) Package (Leake, et al., 1991) may only be used in a transient flow simulation. For steady state flow simulations, this menu item is deactivated.

Removal of ground water by pumping from aquifers may result in compaction of compressible fine-grained beds that are within or adjacent to the aquifers. Compaction of the sediments and resulting land subsidence may be permanent if the head declines result in vertical stresses beyond the previous maximum stress. IBS is designed to simulate storage changes from both elastic and inelastic compaction. In IBS, elastic compaction or expansion is assumed to be proportional to change in head. The constant of proportionality is the product of the skeletal component of elastic specific storage and the thickness of the sediments. Similarly, inelastic compaction is assumed to be proportional to decline in head. The constant of proportionality is the product of the skeletal component of inelastic specific storage and the thickness of the sediments. Within a model time step, IBS appropriately apportions storage changes between elastic and inelastic components based on the relation of simulated head to the previous minimum (preconsolidation or critical) head.

The required input parameters for all input methods are identical and listed below.

1. **All input methods:** Figure 70 shows the dialog box for specifying cell, vertex, or polygon properties.

- **Data Tab**

- **Preconsolidation Head (HC) or Critical Head [L]:**

Preconsolidation head is the previous minimum head value in the aquifer.

For any model cells in which the specified preconsolidation head is greater than the initial hydraulic head, the value

of the preconsolidation head will be set to that of the initial hydraulic head;

When compressible fine-grained sediments are stressed beyond a previous maximum stress (preconsolidation stress), compaction is permanent (inelastic).

- **Elastic Storage Factor (Sfe) [-]:** IBS uses the following equation to calculate the change in the thickness Δb [L] of the interbed (positive for compaction and negative for expansion):

$$\Delta b = -\Delta h \cdot Sfe \quad (18)$$

where Δh [L] is the change in hydraulic head, positive for increase.

- **Inelastic Storage Factor (Sfv) [-]:** IBS uses the following equation to calculate the change in the thickness Δb [L] of the interbed (positive for compaction and negative for expansion):

$$\Delta b = -\Delta h \cdot Sfv \quad (19)$$

- **Starting Compaction (COM) [L]:** Compaction values computed by the IBS package are added to the starting compaction so that stored values of compaction and land subsidence may include previous components. The starting compaction does not affect the calculation of storage changes or resulting compaction.

- **Parameter Number (PEST)** is used to group cells, where the value of the inelastic storage factor is to be estimated by PEST.

- **Options Tab:** The following optional output are available. Check the corresponding box to include the output values in the modflow.interbed file.

- Subsidence
 - Compaction of Individual Layers
 - Preconsolidation Heads

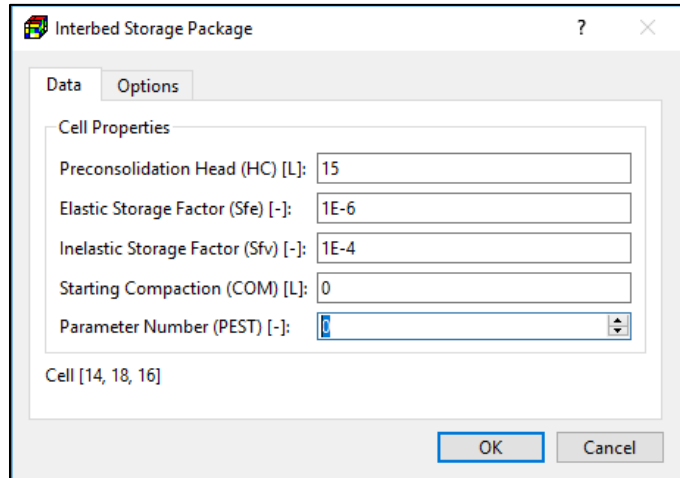


Figure 70. The Interbed Storage Package dialog box

3.9.1.1.9. Lake (LAK3)

The Lake (LAK3)⁴⁶ Package (Merritt, et al., 2000) simulates stationary surface-water bodies (lakes) that are in direct contact with the aquifer. A lake is represented as a volume of space within the model grid which consists of inactive cells extending downward from the upper surface of the grid. Active model grid cells bordering this space, representing the adjacent aquifer, exchange water with the lake at a rate determined by the relative heads and by conductances that are based on grid cell dimensions, hydraulic conductivities of the aquifer material, and user-specified leakage distributions that represent the resistance to flow through the material of the lakebed. Parts of the lake may become “dry” as upper layers of the model are dewatered, with a concomitant reduction in lake surface area, and may subsequently rewet when aquifer heads rise. An empirical approximation has been encoded to simulate the rewetting of a lake that becomes completely dry.

The variations of lake stages are determined by independent water budgets computed for each lake in the model grid. This lake budget process makes the package a simulator of the response of lake stage to hydraulic stresses applied to the aquifer. Implementation of a lake water budget requires input of parameters including those representing the rate of lake atmospheric recharge and evaporation, overland runoff, and the rate of any direct withdrawal from, or augmentation of, the lake volume. The lake/aquifer interaction may be simulated in both transient and steady-state flow conditions, and the user may specify that lake stages be computed explicitly, semi-implicitly, or fully implicitly in transient simulations.

Because lakes with irregular bathymetry can exist as separate pools of water at lower stages, that coalesce to become a single body of water at higher stages, logic was added to the Lake Package to allow the representation of this process as a user option. If this option is selected, a system of linked pools (multiple-lake system) is identified in each time step and stages are equalized based on current relative sublake surface areas.

Lake cells are defined with the cell-by-cell input method. The polyline and polygon input methods are not supported.

1. Cell-by-cell input method: Figure 71 shows the dialog box for specifying cell properties.

- **Parameters Tab**

- **Simulation Settings**

- **Number of Lakes (NLAKES) [-]:** This is the number of lakes to be simulated throughout the entire simulation. Sublakes of multiple-lake systems (see *Options Tab* below) are considered separate lakes for input purposes.

⁴⁶ MODFLOW-2005 will not work if the LAK3 Package is activated and the LAK box of Modflow Output Control (Section 3.9.1.3) is checked. If the LKT Package of MT3D-USGS is not used, uncheck the LAK box. If you need to use the LKT Package, the LAK box must be checked, and you must set MODFLOW version to either MODFLOW-NWT or MODFLOW-OWHM.

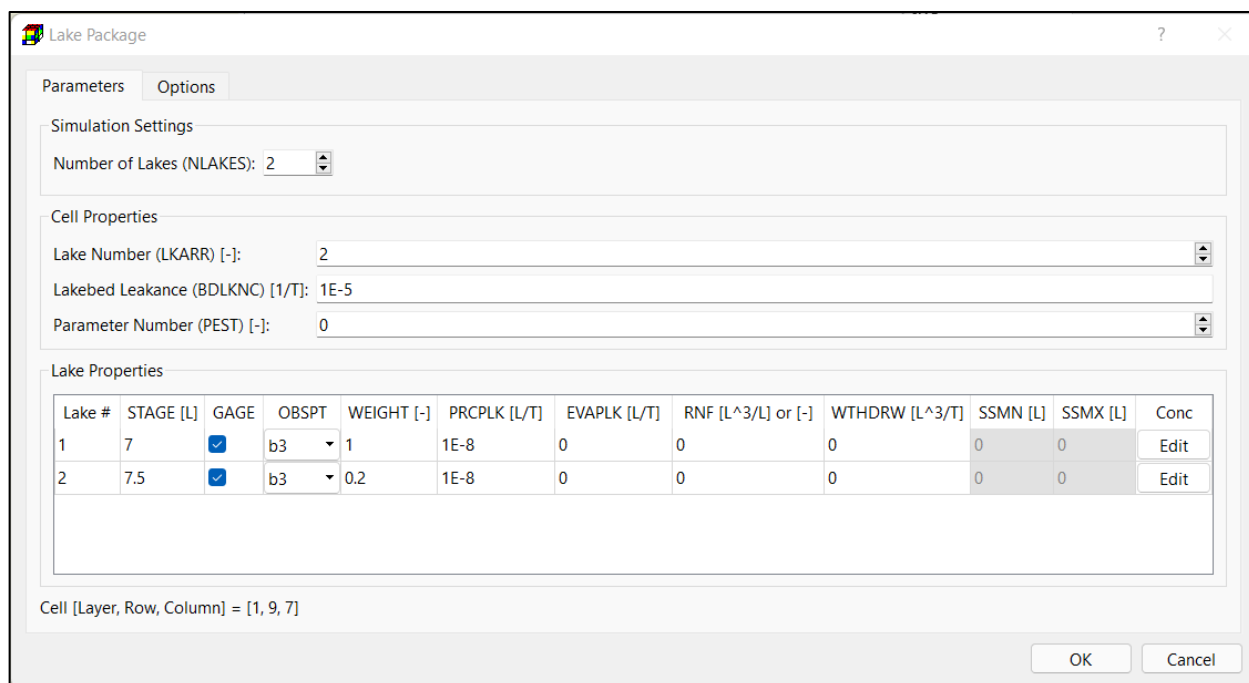


Figure 71. The Lake Package dialog box

- **Cell Properties:** This group define the properties of the lake cell for the current stress period.
 - **Lake Number (LKARR) [-]:** This is the identification number of the lake occupying the grid cell. The lake number must be positive and must not exceed NLAKES. A lake must be in a grid cell that meets the following requirements:
 - The cell must be a no-flow cell (i.e., IBOUND = 0)
 - The cell must not be re-wettable (see Section 3.9.1.1.22).
 - The cell may not be overlain by non-lake cells.
 - **Lakebed Leakance (BDLKNC) [1/T]:** This is the lakebed leakance that will be assigned to lake/aquifer interfaces that occur in the corresponding grid cell.
 - **Parameter Number (PEST) [-]** is used to group cells, where the BDLKNC value is to be estimated by PEST.
- **Lake Properties:** This table define the hydrologic properties of individual lakes for the current stress period.
 - **Lake #:** This is the identification number of a lake.
 - **STAGE [L]:** This is the lake stage at the beginning of the simulation.
 - **GAGE [-]:** If this box is checked, the simulated lake stage will be written by the Gage Package to its output file (modflow.lakegageoutput).
 - **OBSPT [-]:** This dropdown box contains a list of observation points that have stage observation data. If GAGE is checked and an observation point is selected, the calculated lake stage values can be displayed in the Line Chart dialog box (Section 3.9.1.6.4); in addition, PEST can use the observed and calculated lake

stage values for model calibration. See *LAKESTAGEOPT* of the PEST Control Data in Section 3.9.5.2 for available options.

- **PRCPLK [L/T]:** The rate of precipitation per unit area at the surface of the lake.
- **EVAPLK [L/T]:** The rate of evaporation per unit area from the surface of the lake.
- **RNF [L³/L] or [-]:** Overland runoff from an adjacent watershed entering the lake.
 - If $RNF > 0$, it is specified directly as a volumetric rate [L³/T].
 - If $RNF < 0$, its absolute value is used as a multiplier applied to the product of PRCPLK and the surface area of the lake at its full stage.
- **WTHDRW [L³/T]:** The volumetric rate of water removal from the lake by means other than rainfall, evaporation, surface outflow, or groundwater seepage. A negative value indicates augmentation.
- **SSMN [L]:** Minimum stage allowed for the lake in steady-state simulation.
- **SSMX [L]:** Maximum stage allowed for the lake in steady-state simulation.
- **Conc:** Click the *Edit* button to open the Lake Concentration dialog box that allows you to specify the initial concentration values, and concentration values associated with the precipitation and runoff of up to 100 species for the lake.
 - The concentration values specified in the Lake Concentration dialog box are used only by the LKT Package of MT3D-USGS.
 - If the LKT Package is not activated or if you are using MT3DMS or SEAWAT, you may use *Models > MT3DMS/SEAWAT > Package > Sink/Source Mixing (SSM) > Lake (LAK)* to specify the concentration values of the lake water.
- **Options Tab (Figure 72)**
 - **Solution Parameters:** These parameters are defined for the entire simulation.
 - **Time-Weighting Factor (THETA) [-]:** THETA is automatically set to a value of 1.0 for all steady-state stress periods. For transient stress periods,
 - $0.0 \leq THETA \leq 0.5$: Explicit solution for lake stages.
 - $0.5 < THETA < 1.0$: semi-implicit solution for lake stages
 - $THETA = 1.0$: implicit solution for lake stages.
 - **Maximum Number of Iterations (NSSITR) [-]:** Maximum number of iterations for the solution for equilibrium lake stages in each MODFLOW iteration for steady-state aquifer head solution.
 - **Convergence Criterion (SSCNCR) [L]:** Convergence criterion for equilibrium lake stage solution by Newton’s method.
 - **Height of Lake Bottom Undulation (SURFDEPTH) [L]:** SURFDEPTH provides a smooth transition for lakes between dry and wet, and thus reduces numerical oscillations. Values of SURFDEPTH ranging from 0.01 to 0.5 have been used successfully in test simulations.
 - **Multiple-Lake Systems:** This group define the properties of the multiple-lake systems for the current stress period.

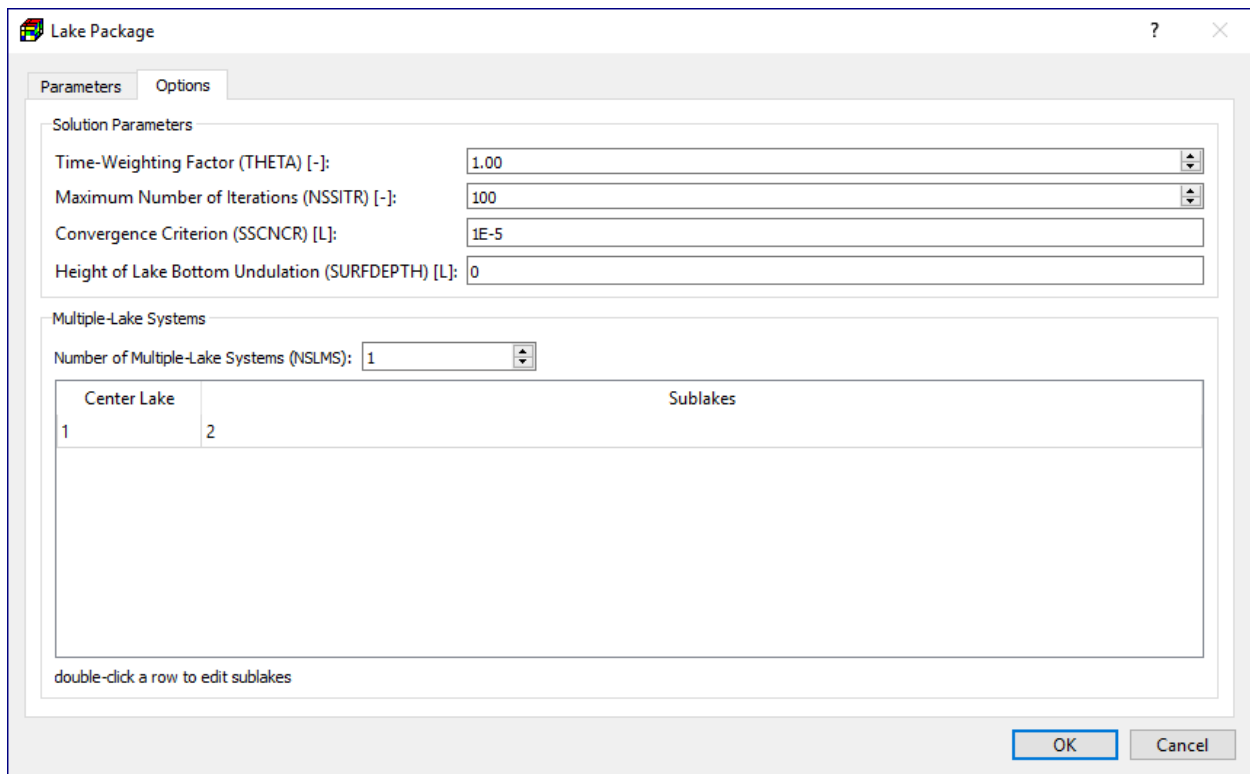


Figure 72. The Options tab of the Lake Package dialog box

- **Number of Multiple-Lake Systems (NSLMS):** The number of multiple-lake systems if coalescing/dividing lakes are to be simulated (only in transient runs). Enter 0 if no multiple-lake systems are to be simulated.
- **Table:** Each row of the table in Figure 72 represents a multiple-lake system. The Center Lake column lists the lake identification number of the center lake of the system; the Sublakes column lists the lake identification numbers of sublakes of the system. Double-click a row to edit the multiple-lake system in the Multiple-Lake System dialog box (Figure 73). The items of the dialog box are as follows:
 - **Center Lake #:** This is the lake identification number of center lake of the multiple-lake system.
 - **Number of Sublakes:** This is the number of sublakes of the multiple-lake system.
 - **Lake #:** This is the lake identification number of a given sublake.

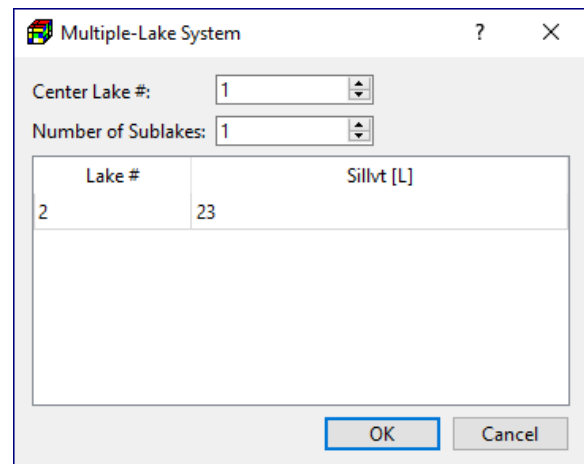


Figure 73. The Multiple-Lake System dialog box

- **Sillvt [L]:** This is the sill elevation, defined as the elevation at which the center lake and the given sublake are connected and have a common state and water budget.

3.9.1.1.10. Multi-Node Well (MNW1)

The Multi-Node Well⁴⁴ (MNW1) Package (Halford, et al., 2002) allows you to simulate wells that extend beyond a single model node (i.e., model cell). The MNW1 Package is superseded by the Multi-Node Well 2 (MNW2) Package (Konikow, et al., 2009). It is recommended to use MNW2 if possible. MNW1 and MNW2 packages cannot both be used in the same simulation.

The MNW1 Package can simulate wells that are completed in multiple aquifers or in a single heterogeneous aquifer, partially penetrating wells, and horizontal wells. Multi-aquifer wells dynamically distribute flow between nodes under pumping, recharging, or unpumped conditions. Variations in intra-borehole flow can be simulated with the MNW1 Package, which is limited by how finely an aquifer system has been discretized vertically. Simulated discharge from single-node and multi-node wells also can be drawdown limited, which is user specified for pumping or recharging conditions. The MNW1 Package can track potential mixes of a water-quality attribute.

A MNW1 well is defined by assigning the following values to a model cell with the cell-by-cell input method. The polyline and polygon input methods are not supported.

1. Cell-by-cell input method: Figure 74 shows the dialog box for specifying cell properties.

- **Parameters Tab**

- **Well Loss Type [-]:** This is the model for well loss. The selected Loss Type applies to all MNW1 wells.
- **Exponent P [-]** is the power (exponent) of the nonlinear discharge component of well loss and is used only when Loss Type is Nonlinear. The specified value of P applies to all MNW1 wells.
- **Reference Stress Period (*kspref*):** The hydraulic head at the beginning of the reference stress period will be used as the reference values for calculating drawdown. See *Limiting Water Level (Hlim)* below for details.
- **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML⁻³]** are associated with the source water. If this box is not checked, the concentration of the source water is assumed to be zero.
- **Well**
 - **Edit Wells:** Click this button to open the MNW1 Wells dialog box that allows you to edit wells.
 - **Well Name (Site) [-]:** Select an existing well to which the following values apply.
 - **Desired Rate (Qdes) [L³T⁻¹]:** The desired injection or extraction rate of the well. Enter a positive value for injection; a negative value for extraction.

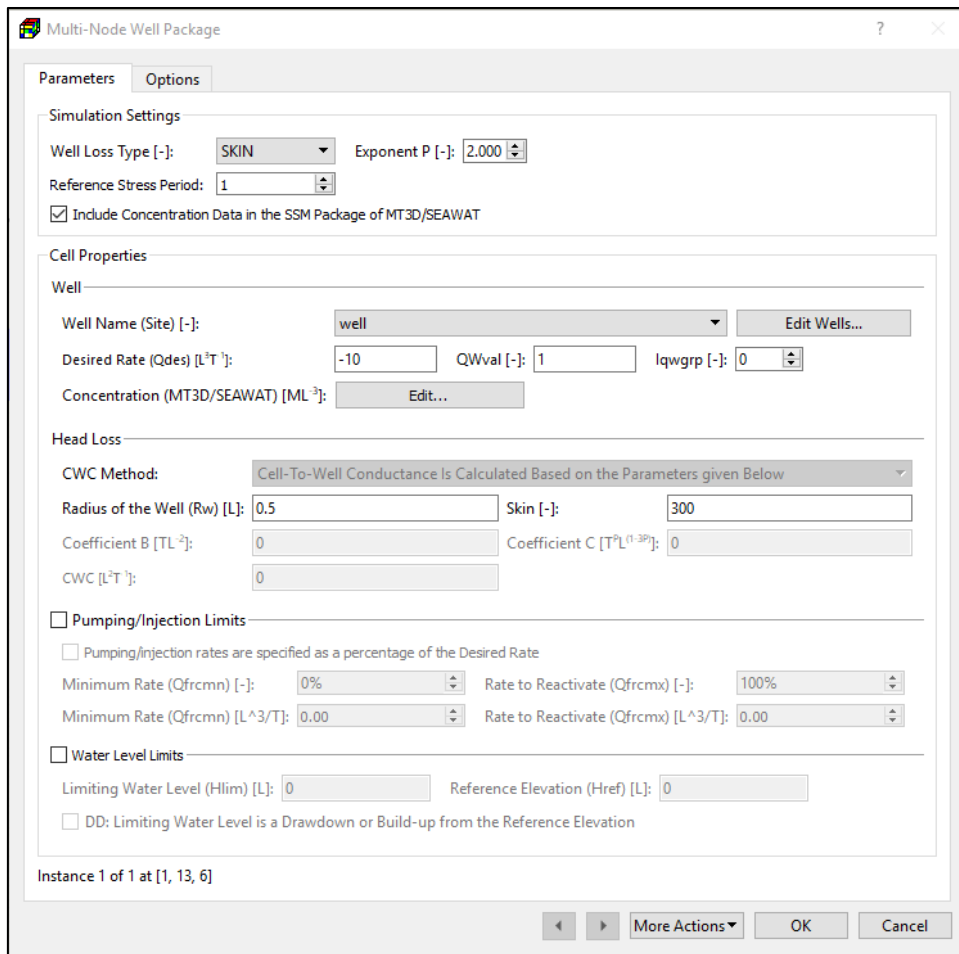


Figure 74. The Multi-Node Well Package dialog box

- **QWval [-]:** Water-quality value that is to be flow-rate averaged amongst wells in the same lqwgrp.
 - **lqwgrp [-]:** Water quality group identifier. Flow-rate averaged water-quality values are reported for wells within the same lqwgrp.
 - **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the injected water concentration of up to 100 species. The entered concentration values are used by the SSM Package of MT3D or SEAWAT.
- **Head Loss**
- **CWC Method:** Specify how the value of Cell-to-Well Conductance (CWC) is determined. For multi-node wells, CWC Method must be "Cell-to-well conductance is calculated based on the parameters given below".
 - **Radius of the Well (Rw) [L]:** Rw is used to calculate the cell-to-well conductance as formulated by (Peaceman, 1983).
 - **Skin [-]** defines the friction losses to the well owing to the screen and to formation damage. This value is used only if the Loss Type is Skin.

- **Coefficient B [TL⁻²]:** This is the coefficient B of the general well-loss equation and is used only when the Loss Type is Linear or Nonlinear. The coefficient B accounts for head losses that occur adjacent to and within the borehole and well.
- **Coefficient C [T^PL^(1-3P)]:** This is the coefficient C accounting for the nonlinear well-loss due to turbulent flow near the well. P is the power (exponent) of the nonlinear discharge component of well loss. The coefficient C is used only when the Loss Type is Nonlinear.
- **CWC [L²T⁻¹]:** This value is the specified cell-to-well conductance.
- **Pumping/Injection Limits**
 - **Limits are specified as a percentage of the Desired Rate:** If this box is checked, pumping/injection limits are specified as a percentage of the *Desired Rate (Qdes)*. If this box is not checked, pumping/injection limits are specified as a rate in the unit of [L³/T].
 - **Minimum Rate (Qfrcmn) [-]:** The minimum rate (as a percentage of *Qdes*) that a well must exceed to remain active.
 - **Rate to Reactivate (Qfrcmx) [-]:** The minimum potential rate (as a percentage of *Qdes*) that must be exceeded to reactivate a well.
 - **Minimum Rate (Qfrcmn) [L³/T]:** The minimum rate that a well must exceed to remain active.
 - **Rate to Reactivate (Qfrcmx) [L³/T]:** The minimum potential rate that must be exceeded to reactivate a well.
- **Water Level Limits**
 - **Limiting Water Level (Hlim) [L]:** this value constrains the pumping/injection rate.
 - If the **DD** box is not checked,
 - a. *Hlim* is the minimum water level for *Qdes* < 0, and
 - b. *Hlim* is the maximum water level for *Qdes* > 0.
 - If the **DD** box is checked,
 - a. [*Href* – *Hlim*] is the minimum water level for *Qdes* < 0, and
 - b. [*Href* + *Hlim*] is the maximum water level for *Qdes* > 0.
 - Reference **Elevation (Href) [L]:** If the specified value of *Href* is greater than the maximum hydraulic head value of the *Reference Stress Period (kspref)*, *Href* is set to the simulated hydraulic head value at the location of the well.
 - **DD: Limiting Water Level Is a Drawdown or Build-up from the Reference Elevation:** the status of this box affects how the limiting water level is defined. See *Limiting Water Level (Hlim)* above for details.
- **Options Tab**
 - **BYNODE:** if this box is checked, a file with the specified **File Name** will be created. This file stores flow rates of each node of the multi-node wells.

- **QSUM:** if this box is checked, a file with the specified **File Name** will be created. This file stores the water level and the sum of flow rates of each multi-node well.
- If both boxes of **BYNODE** and **QSUM** are checked, additional output files will be created for each individual multi-node wells. The filename for a well will be composed of the prefix “MNW1Output.”, the well name, and the suffix “.txt”. For example, if the well name is “Well-A”, the output file will be named as “MNW1Outout.Well-A.txt”. Each of those output files will contain time-series of discharge, water level in well, average water-quality value (based on QWval) of the specific well, as well as the flow rates of all nodes of that well.

2. Polyline and Polygon input methods are not supported.

3.9.1.1.11. Multi-Node Well 2 (MNW2)

The Multi-Node Well 2 (MNW2) Package (Konikow, et al., 2009) allows you to simulate wells that extend beyond a single model node (i.e., model cell). The MNW2 Package is a revised version of the original the Multi-Node Well (MNW1) Package (Halford, et al., 2002). It is recommended to use MNW2 if possible. MNW1 and MNW2 packages cannot both be used in the same simulation. You may also consider converting the input data for the MNW1 Package to the input data for the MNW2 Package by using the utility program *MNW1to2*, that is distributed as a part of MODFLOW 2005 version 1.8 and described in Appendix 3 of the MNW2 document (Konikow, et al., 2009).

The input structure of the MNW2 is more well-centered than MNW1 and allows the user to easily define hydraulic characteristics of each multi-node well. MNW2 also allows calculations of additional head changes due to partial penetration effects, flow into a borehole through a seepage face, changes in well discharge related to changes in lift for a given pump, and intra-borehole flows with a pump intake located at any specified depth within the well. MNW2 also offers an improved capability to simulate nonvertical wells.

MNW2 wells are defined with the cell-by-cell input method. The polyline and polygon input methods are not supported.

1. **Cell-by-cell input method:** Right-click on a model cell to display the Multi-Node Well 2 dialog box (Figure 75).

- **Control Buttons**

- **Pan To:** Click a row in the Wells table, and then click this button to center that well in the middle of the Map Viewer.
- **Zoom To:** Click a row in the Wells table, and then click this button to zoom to that well.
- **Add:** Click this button to add a new well.
- **Delete:** To delete wells, use the combination of drag, shift+click or ctrl+click to select the rows from the Well Table, then click the Delete button to delete the selected rows.

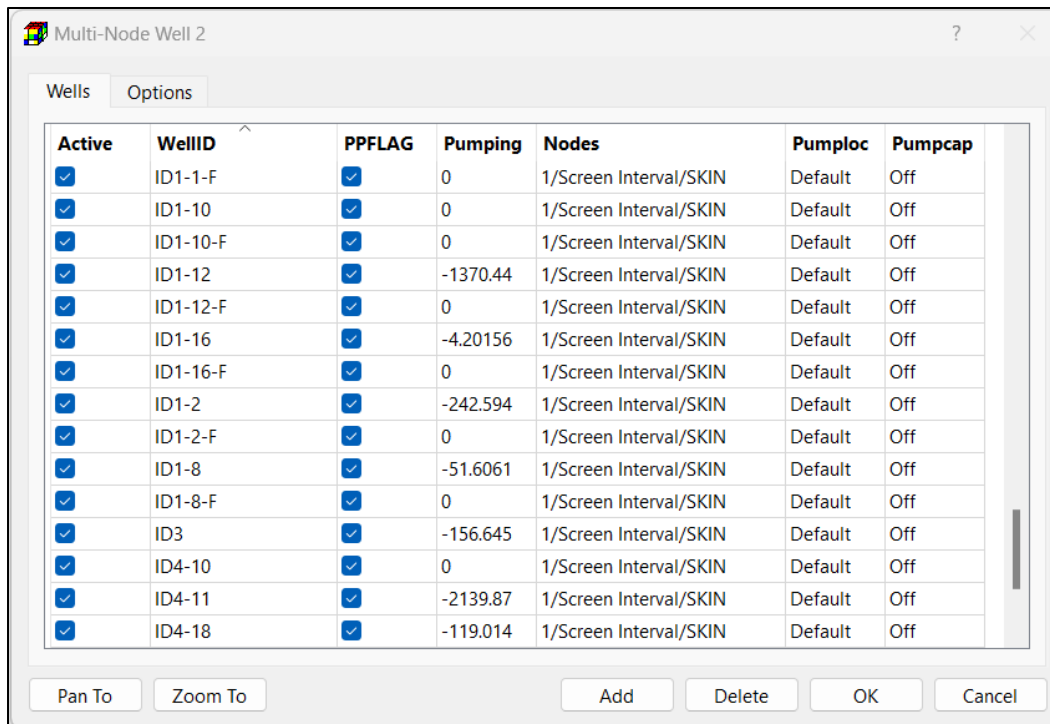


Figure 75. The Multi-Node Well 2 dialog box

- **Wells Table:** The table contains a list of wells. The columns of this table are as follows.
 - **Active:** Check the box to include the well in the simulation. Uncheck to exclude the well from the simulation.
 - **WellID [-]:** This column contains the unique alphanumeric well identifiers. The identifier is limited to 20 characters, including spaces.
 - **PPFLAG [-]:** Check the box to instruct MNW2 package to correct the calculated head in the well for the effect of partial penetration of the well screen in the cell.
 - **Pumping:** The values shown in this column are the applicable pumping rates of the current period. Double-click on a cell in the Pumping column to edit the pumping rates of the respective well. See **Pumping Rates** below for details.
 - **Nodes:** The text strings shown in this column indicate the configuration details of the well nodes. Each text string contains three parts separated by slash (“/”). The first part is the number of nodes, the second part shows whether the nodes are defined by screen intervals or by model layers, the third part shows the loss type. Double-click on a cell in the Nodes column to edit the nodes of the respective well. See **Well Nodes** below for details.
 - **Pumploc:** This column shows how the pump intake location along the borehole is defined. Double-click on a cell in the Pumploc column to edit the pump intake location of the respective well. See **Pump Location** below for details.
 - **Pumpcap:** This column shows whether the pumping rates will be adjusted for changes in the lift with time. Double click a cell in the Pumpcap column to edit the respective well’s pump performance curve and related parameters that are used to adjust pumping rates. See **Pumpcap and Pump Curve** below for details.

- **Options Tab (Figure 76):**
 - **WEL1flag:** If this box is checked, an output file with the specified **File Name** will be created. The flows from every MNW2 node at the end of each stress period are written as single-cell fluxes to that file in the format of the original MODFLOW Well (WEL) Package. Although flow rates for constrained wells can change during a stress period, only flow rates from the last time step of each stress period are reported because the WEL Package is limited to a pumping rate that is constant and uniform in a given well during each stress period. The WEL file will not include wells that were specified using the standard WEL Package.
 - **QSUMflag:** If this box is checked, a file with the specified **File Name** will be created. This file stores the flow rates from each multi-node well for each time step. The QSUM file will include a table of values for all multi-node wells for all times that consists of the sum of all nodal inflows ($Q < 0$) from the aquifer to each well (L^3/T), the sum of all nodal outflows (discharges; $Q > 0$) to the aquifer from each well (L^3/T), the net flow at the wellhead Q_{net} (L^3/T), and the calculated head in the well (L). The modified versions of GSFLOW and MODFLOW-NWT (that are distributed with PM) added two additional columns, Q_{des} and $Q_{net}-Q_{des}$, to the QSUM file. Q_{des} is the user-specified desired pumping rate (L^3/T). These two columns facilitate the comparison of the simulated and specified pumping rates.
 - **BYNDflag:** If this box is checked, an output file with the specified **File Name** will be created. The flows and other information associated with every MNW2 node are written to that file. The BYND file will include a table of values for all nodes of all multi-node wells for all times listing the flow between the node and the aquifer ($Q < 0$ represents flow out of the aquifer into the well; $Q > 0$ represents flow out of the well into the aquifer), and the calculated heads in both the cell and the well.
 - **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the concentration values, that are entered through the Conc [M/L^3] column of the MNW2 Pumping dialog box (Figure 77), are used as the injected water concentration in the SSM Package. If this box is not checked, the injected water concentration is assumed to be zero.

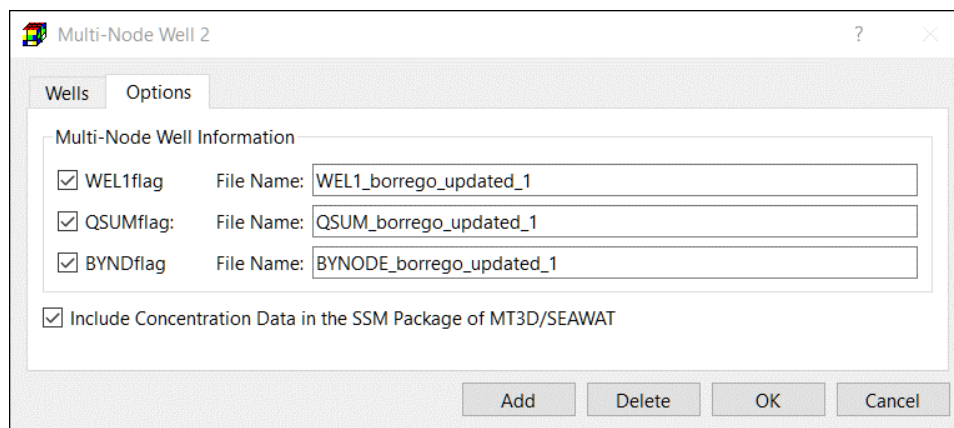


Figure 76. The Options Tab of the Multi-Node Well 2 dialog box

- **Pumping Rates**

You can enter the pumping rates of a well by double-clicking on its Pumping cell in the Multi-Node Well 2 dialog box (Figure 75). This will open the MNW2 Pumping dialog box (Figure 77) with the following settings.

- **WellID:** This is the WellID of the current well.
- **Pumping Constraints (Qlimit):** If this box is checked, the water level (head) in the well will be used to constrain the pumping rate, and relevant parameters (Hlim, Qcut, Qfrcmn, and Qfrcmx) are specified in the table of this dialog box. If this box is not checked, then there are no constraints for this well.
- **Pumping Table:** The table contains the stress periods, and their desired pumping rates, concentration of injection water, and pumping constrain parameters. You can add a row to the table by clicking the Add button. To delete rows, use the mouse to drag and select the rows that you want to delete, then click the Delete button. You can use More Actions to load, save, and clear the table.
 - **Period:** This specifies the stress period number associated with the pumping parameters. If a stress period is missing, the parameters from the previous stress period will be applied. If no previous stress period exists, the pumping rate defaults to zero.
 - **Qdes [L³/T]:** This represents the target volumetric pumping rate at the well. Use negative values for withdrawal and positive values for injection. Qdes should be set to 0 for non-pumping wells. If constraints are applied, the calculated volumetric withdrawal or injection rate may be adjusted within the range of 0 to Qdes. The flow direction (withdrawal or injection) remains unchanged throughout each stress period. When Pumpcap (Figure 75) is on, in the first stress period in which Qdes is specified with a negative value, Qdes represents the maximum operating discharge for the pump. Any different

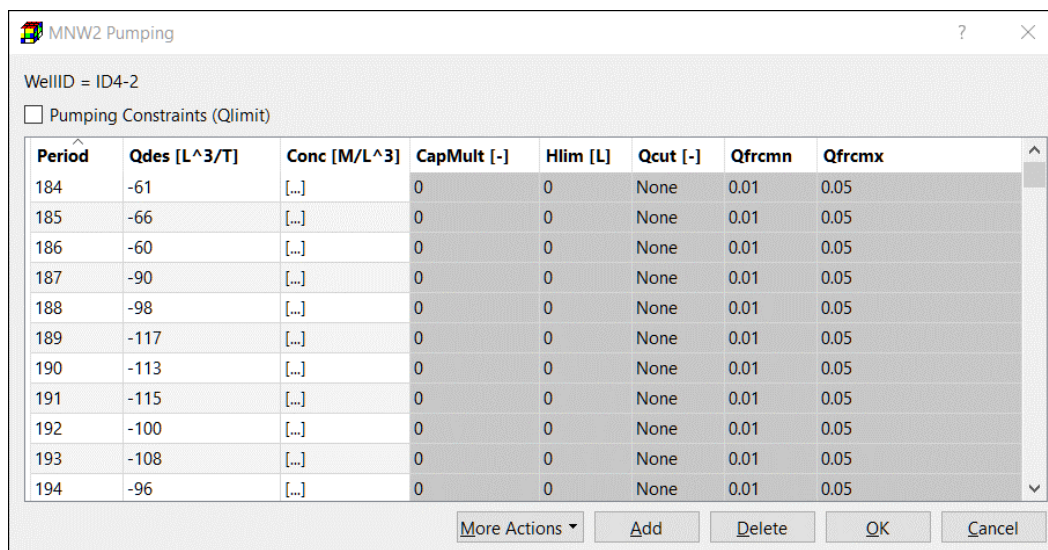


Figure 77. The MNW2 Pumping dialog box

negative values of Q_{des} are ignored in subsequent stress periods in which head-capacity relations are used. If $Q_{des} \geq 0.0$, then pump-capacity adjustments are not applied.

- **Conc [M/L³]**: Double-click a cell to open a Concentration dialog box (Figure 55) that allows you to specify the injected water concentration of up to 100 species. You can either enter individual concentration values or click the *Set Value* button in that dialog box to enter a uniform value for all species. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.
- **CapMult [-]**: This is a multiplier for implementing head-capacity relations (see Pump Performance Curve below) during a given stress period. CapMult is not used and grayed out if Pumpcap (Figure 75) is off.
 - **CapMult ≤ 0** : head-capacity relations are ignored for this stress period.
 - **CapMult > 0** : head-capacity relations are used but adjusted by multiplying the discharge value indicated by the head-capacity curve by the value of CapMult.
- **Hlim [L]** is the limiting water level (head) in the well, which is a minimum for discharging wells and a maximum for injection wells. For example, in a discharging well, when h_{WELL} falls below Hlim, the pumping from the well stops.
- **Qcut [-]** indicates how pumping limits Qfrcmn and Qfrcmx will be specified. The available options are as follows.
 - **Rate**: pumping limits are to be specified as a rate (L³/T)
 - **Fraction**: pumping limits are to be specified as a fraction of the specified pumping rate (Q_{des}).
 - **None**: there is not a minimum pumping rate below which the pump becomes inactive.
- **Qfrcmn** is the minimum pumping rate or fraction of original pumping rate (a choice that depends on Qcut) that a well must exceed to remain active during a stress period. The absolute value of Qfrcmn must be less than the absolute value of Qfrcmx (defined next). Qfrcmn is not used and grayed out if Qcut is set to None.
- **Qfrcmx** is the minimum pumping rate or fraction of original pumping rate that must be exceeded to reactivate a well that had been shut off based on Qfrcmn during a stress period. The absolute value of Qfrcmx must be greater than the absolute value of Qfrcmn. Qfrcmx is not used and grayed out if Qcut is set to None.

- **Well Nodes**

The MNW2 Nodes dialog box (Figure 78) is used to define nodes for the well that is displayed in the dialog box. This dialog box is opened by double-clicking on a cell in the Nodes column of the Multi-Node Well 2 dialog box (Figure 75). The available settings of the dialog box are as follows.

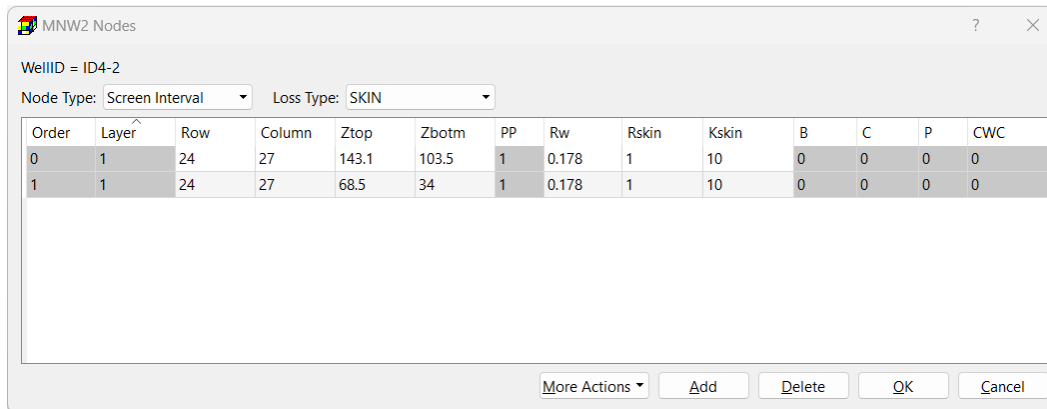


Figure 78. The MNW2 Nodes dialog box

- **Node Type** defines how the well nodes are defined.
 - **By Node:** All nodes are specified by the user in the Node Table of this dialog box (see below) in sequential order from the top to the bottom of the well.
 - **Screen Interval:** for the case of a vertical borehole, this node type allows the user to specify the elevations of the tops and bottoms of well screens in the Node Table (see below) of this dialog box. The MNW2 Package will compute the layers in which the well screens occur, the lengths of the well screens, and the relative vertical position of the well screen within a model layer.
- **Loss Type:** The MNW2 Package uses the following general well-loss equation to calculate the difference between the head in the model cell and the head in the well (the cell-to-well drawdown):

$$h_{well} - h_n = A Q_n + B Q_n + C Q_n^p \quad (20)$$

where h_{well} is the composite head (or water level) in the well (L), n is the index of nodes in a multi-node well, h_n is the head in the n th cell associated with the well (L), Q_n is the flow between the n th cell and the well (L^3/T) (negative for flow out of the aquifer and into the well), A is a linear aquifer-loss coefficient (T/L^2), B is a linear well-loss coefficient (T/L^2), C is a nonlinear well-loss coefficient ($T^p/L^{(3p-1)}$), and p is the power (exponent) of the nonlinear discharge component of well loss. The following loss types are available in MNW2. For more details, refer to Head Loss Terms in (Konikow, et al., 2009).

- **NONE:** there are no well corrections and the head in the well is assumed to equal the head in the cell. This option is only valid for a single-node well. (This is equivalent to using the original WEL Package of MODFLOW but specifying the single-node well within the MNW2 Package enables the use of constraints.)
- **THIEM:** This option allows for only the cell-to-well correction at the well based on the Thiem equation (Thiem, 1906); head in the well is determined as $h_{well} = h_n + A Q_n$, where A is computed as $A = \ln(r_0/r_w) / (2\pi b \sqrt{K_x K_y})$ where r_0 is the effective external radius of a model cell (Peaceman, 1983; Konikow, et al.,

2009), r_w is the user-specified well radius, b is the saturated thickness, and K_x and K_y are the values of hydraulic conductivity in the x- and y-directions. If r_0 is less than r_w , MNW2 sets the value of A and the pumping rate of the corresponding well to zero.

- **SKIN:** This option allows for formation damage or skin corrections at the well; head in the well is determined as $h_{well} = h_n + AQ_n + BQ_n$ where B is determined by the model from the user-specified R_{skin} and K_{skin} . For details, see MNW2 report (Konikow, et al., 2009).
- **GENERAL:** Head in the well is calculated based on the general well-loss equation $h_{well} = h_n + AQ_n + BQ_n + CQ_n^p$. A is computed as $A = \ln(r_0/r_w) / (2\pi b \sqrt{K_x K_y})$. The user must specify R_w , B , C , and P . A value of $P = 2.0$ is suggested if no other data are available (the model allows $1.0 \leq P \leq 3.5$). Entering a value of $C = 0$ will result in a “linear” model in which the value of B is entered directly (rather than entering properties of the skin, as with the SKIN option).
- **SPECIFYcwc:** The well-loss is calculated based on the user specified effective conductance value CWC between the well and the cell representing the aquifer. CWC is equivalent to $[AQ_n + BQ_n + CQ_n^{p-1}]^{-1}$ of the GENERAL well-loss type.

o **Node Table**

You can add a row to the table by clicking the Add button. To delete rows, use the mouse to drag and select the rows that you want to delete, then click the Delete button. You can use More Actions to load, save, and clear the table.

- **Order:** This is used only if *Node Type = By Node*. The model cells (nodes) for the current well are sorted from the top to the bottom based on the ascending Order values, i.e., nodes with smaller Order values are placed on top of the others. The Order values do not need to be sequential.
- **Layer, Row, Column:** These are the layer, row, and column numbers of each model cell (node) for the current well. A particular model cell (node) can be associated with more than one multi-node well. Layer is used only if *Node Type = By Node*. Row and Column should remain the same for all nodes, if *Node Type = Screen Interval* (i.e., vertical well).
- **Ztop, ZBotm:** These are the top and bottom elevations of the screened intervals of a vertical well. These values are used only if *Node Type = Screen Interval*. If an interval partially or fully intersects a model layer, then a node will be defined in that cell. If more than one open interval intersects a particular layer, then a length-weighted average of the cell-to-well conductances will be used to define the well-node characteristics; for purposes of calculating effects of partial penetration, the cumulative length of well screens will be assumed to be centered vertically within the thickness of the cell. If the well is a single-node well by definition of *Loss Type = NONE* and the defined open interval straddles more than one model layer, then the well will be associated with the cell where the center of the open interval exists.

- **PP** is the fraction of partial penetration for this cell. PP is used for a well only if its PPFLAG is checked and *Node Type* is set to Layer.
- **Rw** is the radius of the well. Rw is used if *Loss Type* is set to THIEM, SKIN, or GENERAL.
- **Rskin** and **Kskin**: Rskin is the radius of the outer limit of the skin. Kskin is the hydraulic conductivity of the skin. Rskin and Kskin are used only if *Loss Type* is set to SKIN.
- **B**, **C**, and **P** are the coefficients in the well-loss equation given above. These coefficients are used only if *Loss Type* is set to GENERAL.
- **CWC** is the cell-to-well conductance. CWC is used only if *Loss Type* is set to SPECIFYcwc.

- **Pump Location**

The MNW2 Pumploc dialog box (Figure 79) is used to define the pump intake location along the well borehole. This dialog box is opened by double-clicking on a cell in the Pumploc column of the Multi-Node Well 2 dialog box (Figure 75). The available settings of the dialog box are as follows.

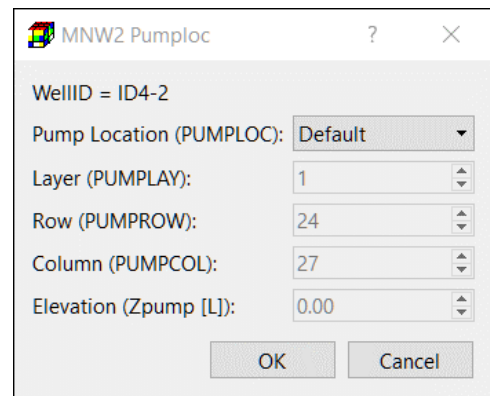


Figure 79. The MNW2 Pumploc dialog box

- **Pump Location (PUMPLOC):**

- **Default:** the intake location (or discharge point for an injection well) is assumed to occur above the first active node associated with the multi-node well (that is, the node closest to the land surface).
- **By Layer:** Use PUMPLAY to define the layer in which the intake (or outflow) is located.
- **By Elevation:** Use Zpump to define the vertical position of the pump intake (or outflow).

- **Layer (PUMPLAY), Row (PUMPROW), Column (PUMPROL):** These are the layer, row, and column numbers, respectively, of the cell (node) in this multi-node well where the pump intake (or outflow) is located. The location should correspond with one of the nodes specified in the MNW2 Nodes dialog box (Figure 78) for this multi-node well. These values are used only if PUMPLOC = By Layer.

- **Pumpcap and Pump Curve**

The MNW2 Pumpcap dialog box (Figure 80) is used to define the pump performance curve and related parameters for the well that is displayed in the dialog box. This dialog box is opened by double-clicking on a cell in the Pumpcap column of the Multi-Node Well 2 dialog box (Figure 75). The available settings of the dialog box are as follows.

- **Adjust the pumping well discharge for changes in the lift (Pumpcap):** The MNW2 Package allows the user to specify a pump curve (head-capacity curve) for each well. When this box is checked, the model will determine the lift that is calculated as the difference between *Hlift* (see below) and the most recent water level in the well. The lift and the pump curve are then used to estimate the discharge from the well for the next iteration.
- **Pump Curve**
 If the Pumpcap box is checked, the user must specify data to approximate the head-capacity curve for the pump. The end points of the applicable curve, representing values of lift corresponding with both zero discharge and the maximum design discharge, must be specified as *LIFTq0* and *LIFTqmax*, respectively. In addition, a minimum of one additional intermediate point on the curve must be specified in the table of the MNW2 Pumpcap dialog box (Figure 80).
 - **Hlift [L]:** This is the reference head (or elevation) corresponding to the discharge point for the well. This is typically at or above the land surface and can be increased to account for additional head loss due to friction in pipes.
 - **HWtol [L]:** If the computed water level in the well changes less than *HWtol* between successive iterations within a time step, the water level is assumed to be stabilized and the estimated discharge based on the lift and the pump curve will be locked for the remainder of that time step. It is recommended to set *HWtol* to a value approximately one or two orders of magnitude larger than the value of the head change criterion for convergence of the selected solver (see Section 3.9.1.2).
 - **LIFTq0 [L]:** This is the value of the maximum lift of the pump. If the calculated lift equals or exceeds this value, then the pump is shut off and discharge from the well ceases.

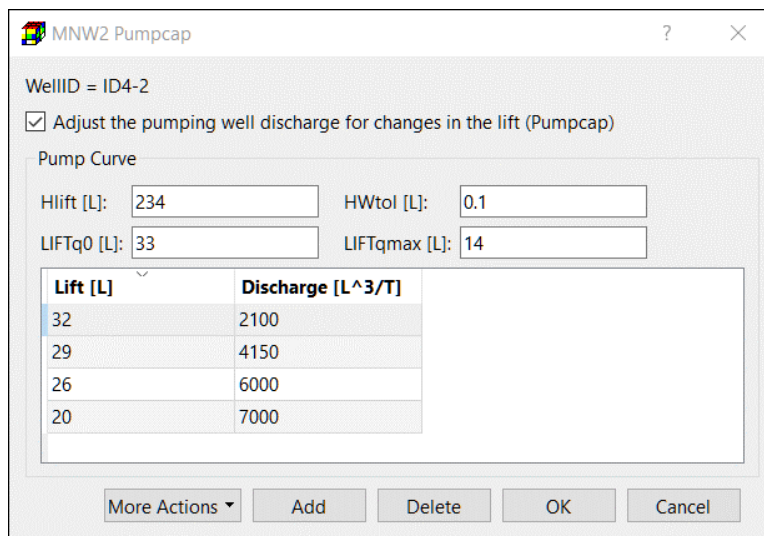


Figure 80. The MNW2 Pumpcap dialog box

- **LIFTqmax [L]:** This is the value of lift corresponding to the maximum discharge rate for the pump. If the calculated lift is less than or equal to *LIFTqmax*, then the pump will operate at its design capacity, assumed to equal the user-specified value of *Qdes* given in the MNW2 Pumping dialog box (Figure 77). *LIFTqmax* will be associated with the value of *Qdes* in the first stress period in which *Qdes* for the well is less than 0.0.
- **Pump Curve Table**
You can add a row to the table by clicking the Add button. To delete rows, use the mouse to drag and select the rows that you want to delete, then click the Delete button. You can use More Actions to load, save, and clear the table.
 - **Lift [L]:** This is a value of lift that corresponds to a known value of discharge for the given pump, specified in the second column of the table.
 - **Discharge [L³/T]:** This is the value of discharge corresponding to the lift specified in the first column of the table. Sign (positive or negative) is ignored.

3.9.1.1.12. Recharge (RCH)

The Recharge (RCH) Package is designed to simulate distributed recharge to the ground-water system. The RCH Package does not allow for recharge to occur simultaneously at multiple depths in the same vertical column because natural recharge enters the ground-water system at the top, and therefore a value of the recharge flux *RECH* [LT⁻¹] is assigned to a cell in each vertical column of model cells. The recharge rate to the groundwater system Q_{rch} [L³T⁻¹] is calculated by

$$Q_{rch} = RECH \cdot Area_c \quad (21)$$

where

RECH [LT⁻¹] is the recharge flux at the cell, and

Area_c is the area of the cell.

The required input parameters for all input methods are identical and listed below. Although these parameters are to be assigned to each vertical column of model cells, you may move between layers within the Data Editor to examine the grid configuration in each layer.

1. **All input methods:** Figure 81 shows the dialog box for specifying cell, vertex, or polygon properties.

- **Data Tab**

- **Recharge Option (NRCHOP) [-]:** Since Q_{rch} is calculated for only one cell in each vertical column of model cells, three options are available for specifying the cell where Q_{rch} is applied to:
 - Recharge is only applied to the top layer.
 - Vertical distribution of recharge is specified in IRCH
 - Recharge is applied to the highest variable-head cell in each vertical column

- **Set the recharge flux array multiplier of the current period as a PEST parameter:** Check this box to designate the recharge flux array multiplier of the current period as an adjustable parameter. The name of an adjustable parameter is RECM_p, where p is the current stress period number. This can be used in conjunction with *Parameter Number (PEST) [-]* below.
- **Recharge Flux (RECH) [LT⁻¹]:** This is the recharge flux of the current stress period.
- **Layer Indicator (IRCH) [-]:** This indicates the layer from which Q_{rch} is added to the groundwater system.
- **Parameter Number (PEST) [-]** is used to group cells, where the RECH value is to be estimated by PEST.
- **Options Tab:** This tab contains the following options. You can check the options to use the respective data of the first stress period for the entire simulation even if the data of other stress periods have been specified.
 - Use Recharge Flux (RECH) of the first stress period for the entire simulation
 - Use Layer Indicator (IRCH) of the first stress period for the entire simulation

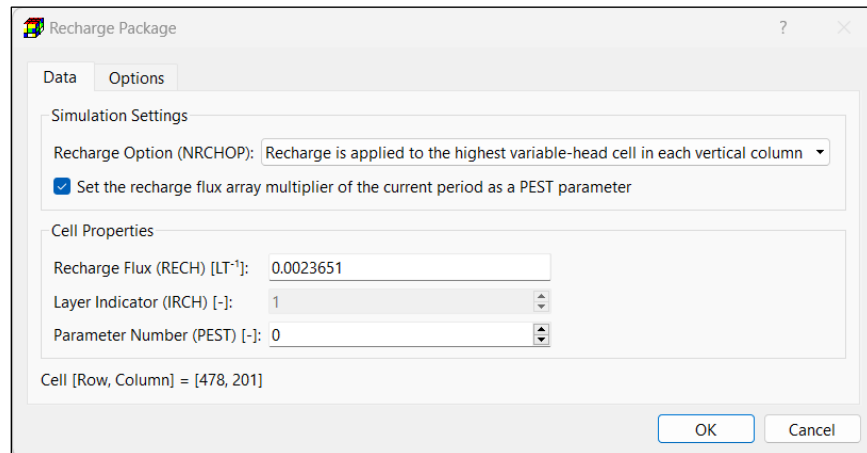


Figure 81. The Recharge Package dialog box

3.9.1.1.13. River (RIV)

The purpose of the River (RIV) Package is to simulate the effects of flow between surface-water features and groundwater systems. Rivers and streams contribute water to or drain water from the groundwater system, depending on the head gradient between the river and the groundwater regime.

A river is implemented by dividing into reaches so that each reach is completely contained in a single model cell. Multiple reaches may be assigned to a model cell. For each reach, the flow rate Q_{riv} [L³T⁻¹] between the reach and the model cell is calculated by the following equations.

$$Q_{riv} = C_{riv} \cdot (h_{riv} - h) \quad \text{if } h > B_{riv} \quad (22)$$

$$Q_{riv} = C_{riv} \cdot (h_{riv} - B_{riv}) \quad \text{if } h < B_{riv} \quad (23)$$

where

Q_{riv} [L^3T^{-1}] is the flow rate between the reach and the model cell (i.e., the groundwater system). A positive value means that water flows from the river to aquifer,

h [L] is the calculated hydraulic head in the cell containing the reach,

h_{riv} [L] is the head in the reach,

B_{riv} [L] is the elevation of the riverbed bottom of the reach, and

C_{riv} [L^2T^{-1}] is the riverbed conductance of the reach.

The required input parameters are listed below.

1. **All input methods:** The data entry dialog boxes of all input methods include a Simulation Settings group that contains the following option.

- **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML^{-3}]** are used as source water concentration. If this box is not checked, the concentration of the source water is assumed to be zero.

2. **Cell-by-cell input method:** Figure 82 shows the dialog box for specifying one or more reaches to a model cell. For each reach, the following properties need to be assigned.

- **Riverbed conductance C_{riv} [L^2T^{-1}]** relates flow rate to the head differences between the reach and the aquifer. The value of C_{riv} depends on the material and characteristics of the riverbed and the immediate environment.
- **River Stage h_{riv} [L]** is the hydraulic head of the reach.
- **Elevation of the Riverbed Bottom B_{riv} [L]** is the elevation of the bottom of the riverbed.
- **Parameter Number (PEST)** is used to group cells, where the C_{riv} value is to be estimated by PEST.
- **Thickness of Riverbed Sediments (SEAWAT) [L]:** This value is used by SEAWAT to appropriately include the effects of variable-density river leakage and is used only if RBDTHK (in Figure 119) is set as *User-Specified in the RIV Package*.
- **River Water Density (SEAWAT) [ML^{-3}]** is the water density of the river. This

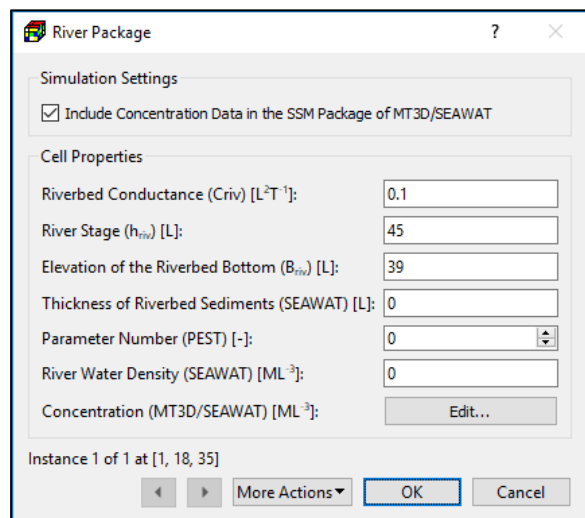


Figure 82. The River Package dialog box for specifying cell properties

is used by SEAWAT only if it is running in the uncoupled mode (i.e., the density-effect of all species are turned off, see Section 3.9.3.1) and RIVDEN (in Figure 119) is set as *User-Specified in the RIV Package*.

- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species to the river. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

3. Polyline input method: Figure 83 shows a dialog box for specify data to one or more vertices of a polyline. The specified vertex data may be interpolated to model cells following the rules described in Section 3.4.2.2 and the rules below.

- **Simulation Settings**

- **Auto Layer Number**

- If this box is checked, the layer number of a reach along the polyline is determined by comparing its elevation of the riverbed bottom (B_{riv}) with the top/bottom elevations of the cells in the same vertical model column as the reach.

- if B_{riv} is higher than the top elevation of the highest cell, the reach is assigned to the cell in Layer 1.
 - if B_{riv} is lower than the bottom elevation of the lowest cell, the reach is assigned to the cell in Layer NLAY.
 - If B_{riv} lies between top and bottom elevations of a cell, the reach is assigned to that cell.

- If this box is not checked, the reaches along the polyline are assigned to the layer where the polyline is located.

- **Vertex Properties:** The data of this group are assigned to one or more vertices of a polyline. These vertex data may be interpolated to model cells following the rules described in Section 3.4.2.2 and the rules below.

- **Riverbed conductance per unit length C_{rivl} [LT⁻¹]** is interpolated to the cells along the polyline. For each of those cells, the interpolated cell value C_{rivl} is multiplied with the length of the river within the cell to obtain the riverbed conductance C_{riv} :

$$C_{riv} = C_{rivl} \cdot L_c \quad (24)$$

where

C_{rivl} is the riverbed conductance per unit length at a given cell, and

L_c is the length of the river within the cell.

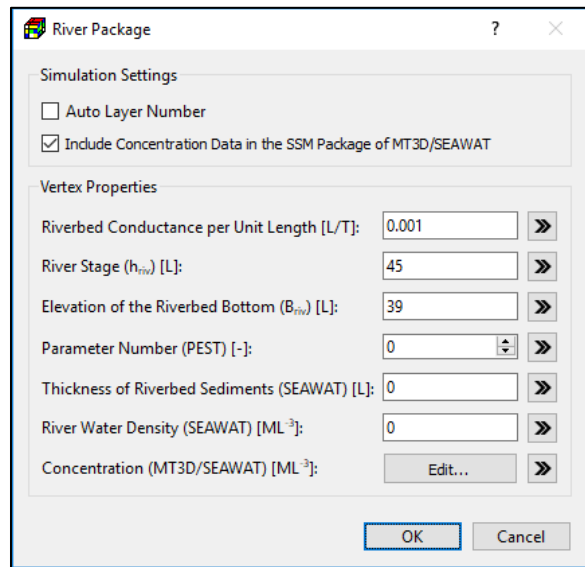


Figure 83. The River Package dialog box for specifying vertex properties

- **All other parameters:** Refer to the cell-by-cell input method above for the descriptions of these parameters.

4. **Polygon input method:** Figure 84 shows a dialog box for specify the following data to a polygon. These data may be applied to model cells following the rules described in Section 3.4.2.3 and additional rules below.

- **Simulation Settings:** See the description for the polyline input method above.
- **Polygon Properties**

- **Riverbed conductance per unit area C_{riva} [T^{-1}].** For each cell within the polygon, the specified value (C_{riva}) is multiplied with the cell area to obtain the riverbed conductance C_{riv} :

$$C_{riv} = C_{riva} \cdot Area_c \quad (25)$$

where

C_{riva} is the riverbed conductance per unit area, and
 $Area_c$ is the area of the cell.

- **The other parameters:** Refer to the cell-by-cell input method above for the descriptions of these parameters.

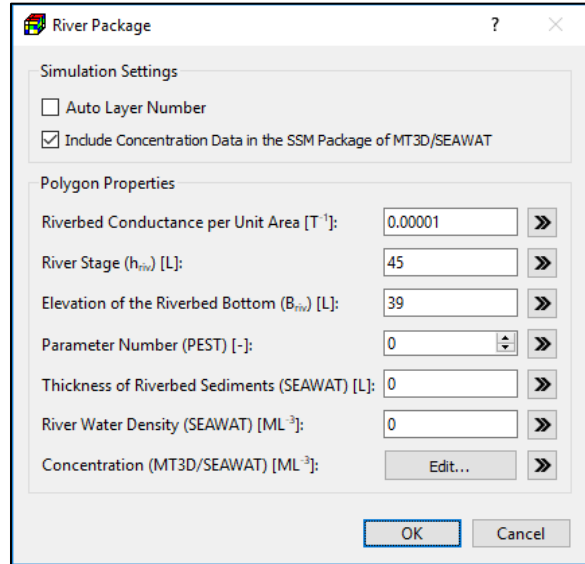


Figure 84. The River Package dialog box for specifying polygon properties

3.9.1.1.14. Reservoir (RES1)

The Reservoir (RES1) Package (Fenske, et al., 1996) simulates leakage between a reservoir and an underlying groundwater system as the reservoir area expands and contracts in response to changes in reservoir stage. The Reservoir Package also automates the process of specifying head-dependent boundary cells during the simulation. For each time step, all cells for which the reservoir stage is above the reservoir land-surface elevation are simulated as active reservoir cells. Leakage between the reservoir and the underlying groundwater system is simulated for each active reservoir cell by multiplying the difference between the reservoir stage and the groundwater head by the hydraulic conductance of the reservoir bed. The package uses linear interpolation to calculate reservoir stage for each time step within a stress period.

The flow rate Q_{res} [L^3T^{-1}] between the reservoir and the underlying groundwater system is simulated for each vertical column of model cells corresponding to the inundated area of the reservoir with the following equations.

$$Q_{res} = C_{res} \cdot (h_{res} - h) \quad \text{if } h > B_{res} - Rbthck \quad (26)$$

$$Q_{res} = C_{res} \cdot (h_{res} - B_{res}) \quad \text{if } h < B_{res} - Rbthck \quad (27)$$

where

Q_{res} [L^3T^{-1}] is the flow rate between the reservoir and the model cell (i.e., the groundwater system) to which the reservoir is connected (see connection option below). A positive value means that water flows from the reservoir to the groundwater system,

C_{res} [L^2T^{-1}] is the reservoir-bed conductance,

h_{res} [L] is the reservoir stage,

h [L] is the calculated hydraulic head in the connected model cell,

B_{res} [L] is the land surface elevation (i.e., elevation of the top of reservoir-bed), and

$Rbthck$ [L] is the thickness of the reservoir-bed.

The reservoir-bed conductance C_{res} [L^2T^{-1}] is calculated by

$$C_{res} = HC_{res} \cdot Area_c / Rbthck \quad (28)$$

where

HC_{res} [LT^{-1}] is the vertical hydraulic conductivity of the reservoir-bed sediments, and

$Area_c$ is the area of the connected cell.

Inherent in the simulation of reservoirs is that the reservoir only partially penetrates an active model cell. If the reservoir fully penetrates a cell, it is assumed that the cell will be specified as no flow and that reservoir flow will be simulated in a lower cell. Thus, water exchange between the ground-water system and the reservoir takes place across the bottom of the reservoir. Refer to p. 4 of (Fenske, et al., 1996) for details.

A reservoir is defined by assigning the following values to each vertical column of model cells with the cell-by-cell or polygon input methods. The polyline input method is not supported. Figure 85 shows the dialog box for specifying cell properties. Although these parameters are to be assigned to each vertical column of model cells, you may move between layers within the Data Editor to examine the grid configuration in each layer.

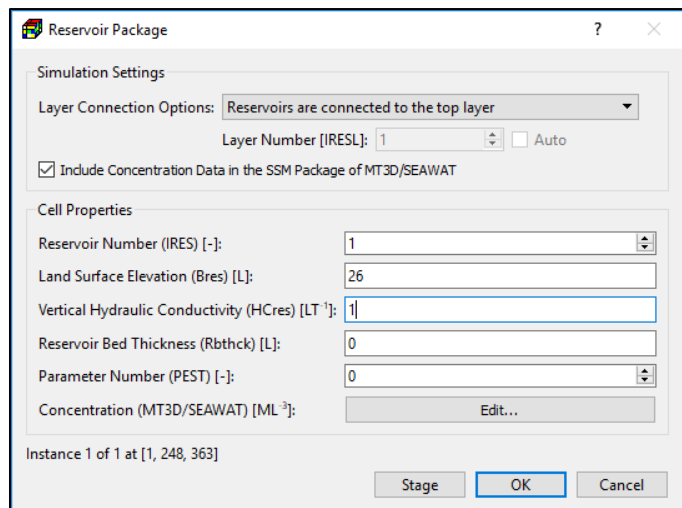


Figure 85. The Reservoir Package dialog box

1. All supported input methods

- **Layer Connection Options:** Define how and where the reservoir leakage is connected to the groundwater system. The connection options apply to all reservoirs. The following options are available.
 - Reservoirs are connected to the top layer
 - Reservoirs are connected to the highest variable-head cells (of each vertical column of model cells.)
 - Reservoirs are connected to the layers defined by IRESL.
- **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML⁻³]** are used as source water concentration. If this box is not checked, the concentration of the source water is assumed to be zero.
- **Layer Number (IRESL) [-]:** Defines the layer to which the current reservoir cell is connected. Required only if the third connection option is selected.
- **Reservoir Number (IRES) [-]:** The number of the reservoir to which the vertical column of cells is connected. A value of zero or less indicates the absence of reservoirs at the corresponding cell location. Values greater than zero indicate that the cell is part of the corresponding reservoir.
- **Land Surface Elevation (Bres) [L]:** The land surface elevation within the specified area of potential inundation for each reservoir is typically defined by the average land surface elevation of individual cells within the area. At cells in which reservoir stage exceeds Bres, the reservoir boundary is activated. Similarly, wherever reservoir stage is less than Bres, the reservoir boundary is not activated.
- **Vertical Hydraulic Conductivity (HCres) [LT⁻¹]:** This is the vertical hydraulic conductivity of the reservoir bed overlying the current cell.
- **Reservoir Bed Thickness (Rbthck) [L]:** This is the thickness of the reservoir-bed sediments overlying the current cell.
- **Parameter Number (PEST)** is used to group cells, where the HCres value is to be estimated by PEST.
- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

- **Stage:** Click this button to open the Reservoir Stage Time Series dialog box (Figure 86) for specifying the water table elevations of reservoirs.

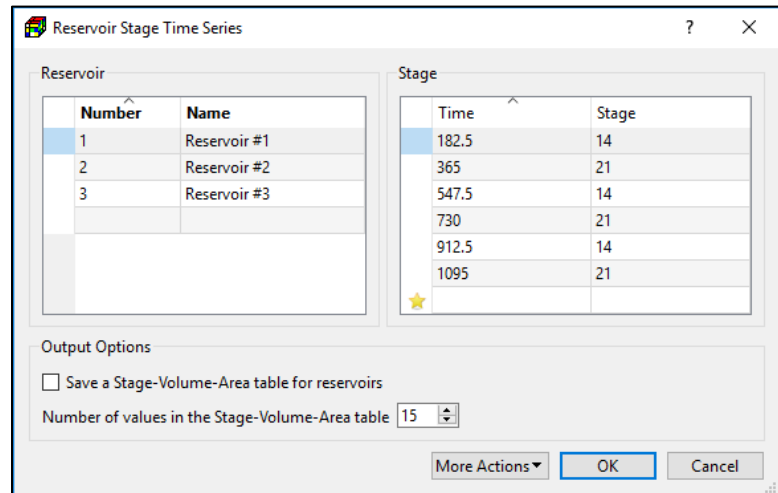


Figure 86. The Reservoir Stage Time Series dialog box

- **Reservoir table:** Select a reservoir number (a row) from the Reservoir table. The reservoir number is corresponding to IRES (see above).

You can optionally assign the names of the reservoirs to the Name column.

- **Stage Table:** Enter the observation time and the corresponding stage to the Stage table. The observation time is expressed in the model time unit and is measured from the start of the model simulation. The Reservoir package requires the input of the starting and ending stages for each stress period. The stage values for each stress period are obtained by linear interpolation using the values specified in the Stage table. If the starting time of a stress period is earlier than the earliest observation time in the table, the earliest observed stage is used as the starting stage for that stress period. Similarly, if the end time of a stress period is beyond the latest observation time, the latest observed stage is used.
- **Output Option**
 - **Save a Stage-Volume-Area table for reservoirs:** If this option is checked, reservoir stage, area, and volume (at the end of each time step) will be printed to the Run Listing File of MODFLOW.
 - **Number of values in the stage-volume-area table:** This is the number of values used to print the Stage-Volume-Area table. First and last stage value are minimum and maximum elevations within area of potential inundation. A value of 15 or greater is recommended for detailed representation of stage-volume and stage-area relations.

3.9.1.1.15. Stream (STR)

The Stream (STR) Package (Prudic, 1989) is designed to account for flow in streams and to simulate the interaction between streams and groundwater. This package was originally called the Stream-flow Routing package and is renamed to the Stream Package in MODFLOW-2005 to avoid the naming conflict with the Streamflow-Routing (SFR1) Package⁴⁷ (Prudic, et al., 2004) and

⁴⁷ SFR1 is not supported by PM

the SFR2 Package (Niswonger, et al., 2006). The STR, SFR1, and SFR2 packages should not be used in a model at the same time.

The STR Package is not a true surface water flow model, but rather is a bookkeeping program that tracks the flow in one or more streams which interact with groundwater. The STR Package limits the amount of groundwater recharge to the available streamflow. It permits two or more streams to merge into one with flow in the merged stream equal to the sum of the tributary flows. The STR Package also permits diversions from streams. The groundwater flow model with the STR Package has an advantage over the analytical solution in simulating the interaction between aquifer and stream because it can be used to simulate complex systems that cannot be readily solved analytically. The STR Package does not include a time function for streamflow but rather streamflow entering the modeled area is assumed to be instantly available to downstream reaches during each stress period. This assumption is generally reasonable because of the relatively slow rate of groundwater flow. Another assumption is that leakage between streams and aquifers is instantaneous. This assumption may not be reasonable if the streams and aquifers are separated by a thick unsaturated zone.

The STR Package divides streams into segments and reaches. Each reach corresponds to a model cell. A segment consists of a group of reaches connected in downstream order. Streamflow is accounted for by specifying flow for the first reach in each segment, and then computing streamflow to adjacent downstream reaches in each segment as inflow in the upstream reach plus or minus leakage from or to the aquifer in the upstream reach. Streamflow into a segment that is formed from tributary streams is computed by adding the outflows from the last reach in each of the specified tributary segments. If a segment is a diversion, then the specified flow into the first reach of the segment is subtracted from flow in the main stream. However, if the specified flow of the diversion is greater than the flow out of the segment from which flow is to be diverted then no flow is diverted from that segment.

Multiple reach instances may be assigned to a model cell. For each reach, the flow rate Q_{str} [L^3T^{-1}] between the reach and the model cell is calculated by the following equations.

$$Q_{str} = C_{str} \cdot (h_{str} - h) \quad \text{if } h > B_{str} \quad (29)$$

$$Q_{str} = C_{str} \cdot (h_{str} - B_{str}) \quad \text{if } h \leq B_{str} \quad (30)$$

where

Q_{str} [L^3T^{-1}] is the flow rate between the reach and the model cell (i.e., the groundwater system). A positive value means that water flows from the reach to aquifer,

C_{str} [L^2T^{-1}] is the stream conductance.

h [L] is the calculated hydraulic head in the cell containing the reach,

h_{str} [L] is the stream stage (i.e., hydraulic head in the reach), and

B_{str} [L] is the elevation of the bottom of streambed sediments.

The cell-by-cell or polyline input methods may be used to specify the following stream data. The polygon input method is not supported. The polyline input method is recommended as it can assign the reach numbers in the required sequential order automatically.

1. **Cell-by-cell input method:** Figure 87 shows the dialog box for specifying reach properties.

- **Calculate Stream Stage in Reaches:** If this box is checked the stream stage in each reach is calculated based on Manning's formula under the assumption of a rectangular stream channel. This option applies to the entire STR package.
- **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML⁻³]** are used as source water concentration. If this box is not checked, the concentration of the source water is assumed to be zero.
- **Segment (Cell Properties)**

Figure 87. The Stream Package dialog box for specifying cell properties

- **Segment Number [-]:** The number of the segment to which the current reach belongs. Enter 0 to remove the cell from the segment.
- **Inflow to this Segment [L³T⁻¹]:** The streamflow entering the segment. Enter -1, if inflow into this segment is the sum of outflow from its tributary segments. If a segment is a diversion and the *Simulate diversions from segments* box (see below) is checked, then the specified inflow is subtracted from flow in the main stream. However, if the specified inflow of the diversion is greater than the flow out of the segment from which flow is to be diverted, then no flow is diverted from that segment.

- **Reach (Cell Properties):**

- **Reach Number [-]:** This is a sequential number in a segment that begins with one for the farthest upstream reach and continues in downstream order to the last reach in the segment.
- **Stream Stage (h_{str}) [L]** is the hydraulic head in the stream. Stream Stage is not used if "Calculate Stream Stage in Reaches" is checked.
- **Streambed Conductance (C_{str}) [L^2T^{-1}]:** The value describes the head loss between the stream and the aquifer. It depends on the material and characteristics of the streambed sediments and the immediate environment. The value of C_{str} can be estimated based on the following equation:

$$C_{str} = \frac{K_{str} \cdot L \cdot W_{str}}{T_{str} - B_{str}} \quad (31)$$

where

K_{str} [LT^{-1}] is the hydraulic conductivity of the streambed,

L [L] is the length of the stream reach within the cell,

W_{str} [L] is the width of the stream channel,

T_{str} [L] is the elevation of the top of the streambed sediments, and

B_{str} [L] is the elevation of the bottom of the streambed sediments.

- **Elevation of Streambed Top (T_{str}) [L]** is the elevation of the top of the streambed sediments.
- **Elevation of Streambed Bottom (B_{str}) [L]** is the elevation of the bottom of the streambed sediments.
- **Width of the Stream Channel (W_{str}) [L], Slope of the Stream Channel (S_{str}) [-], and Manning's Roughness Coefficient (n) [$TL^{-1/3}$]:** These parameters are used only when the box *Calculate Stream Stage in Reaches* is checked. To obtain the stream stage, the stream water depth (d_{str}) is calculated using the Manning's equation under the assumption of a rectangular stream channel. The calculated water depth is added to T_{str} to get the stream stage. The Manning's equation for a stream channel is

$$Q = A \cdot R^{2/3} \cdot S_{str}^{1/2} / n \quad (32)$$

where

Q [L^3T^{-1}] is the calculated stream discharge,

A [L^2] is the cross-sectional area of flow,

R [L] is the hydraulic radius, which is defined as A divided by the wetted perimeter,

S_{str} [-] is the slope of the stream channel, and

n [$TL^{-1/3}$] is the Manning's roughness coefficient.

The unit for Manning's roughness coefficient (n) is often left off, as n is usually given in SI units, for example [$s/m^{1/3}$]. If other units are used in the model, you should convert the value of n accordingly and enter the converted value to the model. For example, if your model uses [day] and [ft] for the time and length units, the following conversion applies

$$n = 1 \left[\frac{s}{m^3} \right] = \frac{1}{86400 \times 3.283^3} \left[\frac{day}{ft^3} \right] = \frac{1}{128383} \left[\frac{day}{ft^3} \right] \quad (33)$$

Some conversion examples for other units are given below

$$n = 1 \left[\frac{s}{m^3} \right] = \frac{1}{86400} \left[\frac{day}{m^3} \right] = \frac{1}{1.486} \left[\frac{s}{ft^3} \right] \quad (34)$$

- **Parameter Number (PEST) [-]** is used to group cells, where the C_{str} value is to be estimated by PEST.
- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.
- **Structure:** Click this button to open the Stream Structure dialog box (Figure 88) that defines the structure of the stream system as shown in Figure 89. The table of dialog box indicates that segment 2 is diverted from segment 1, segment 1 is a tributary segment of segment 3, and segments 2 and 4 are tributary segments of segment 5.

- **Simulate diversions from segments:** Check this box to simulate diversions from segments.

- **Table:** Each row in the table represents a stream segment that is diverted from an upstream segment or has tributary segment(s).

Segment #	1	2	3	4	5	6	7	8	9	10	lUpSeg
1	0	0	0	0	0	0	0	0	0	0	0
2	0	0	0	0	0	0	0	0	0	0	1
3	1	0	0	0	0	0	0	0	0	0	0
4	0	0	0	0	0	0	0	0	0	0	0
5	2	4	0	0	0	0	0	0	0	0	0

- **Segment #** is the number of the segment.
- **Columns 1 to 10:** Each segment can have up to 10 tributary segments and their segment numbers are specified in the *respective columns*.
- **lupseg** is the number of the upstream segment from which water is diverted. For a segment that is not a diversion, *lupseg* must be specified as zero. *lupseg* is used only when the *Simulate diversions from segments* box is checked.

- **Add:** Click this button to add a row to the Table.
- **Delete:** Click this button to delete the selected row(s) from the Table.

2. Polyline input method:

Figure 90 shows the dialog box for specifying vertex/polyline properties. Once the vertex data are specified, they may be interpolated and applied to model cells following the rules described in Section 3.4.2.2.

- **Simulation Settings:** Same as the cell-by-cell input method.
- **Segment (Polyline Properties)**

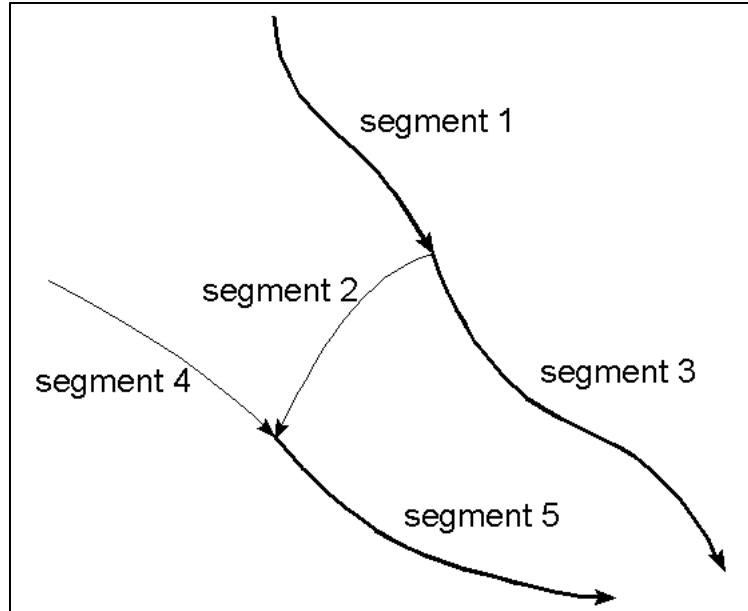


Figure 89. The stream system defined by the table in Figure 88

- **Auto Layer Number**

- If this box is checked, the layer

number of a stream reach along the polyline is determined by comparing the elevation of the streambed bottom B_{str} of the reach with the top/bottom elevations of the cells in the same vertical model column as the reach.

- if B_{str} is higher than the top elevation of the highest cell, the reach is assigned to the cell in Layer 1.
- if B_{str} is lower than the bottom elevation of the lowest cell, the reach is assigned to the cell in Layer NLAY.
- If B_{str} lies between top and bottom elevations of a cell, the reach is assigned to that cell.

- If this box is not checked, the reaches along the polyline are assigned to the layer where the polyline is located.

- **Segment Number [-]:** The number of the segment to which the reaches along the polyline belong. Prior to applying a polyline to model cells, all existing reaches of the given segment number will be removed. Reach numbers will be assigned to the reaches along the polyline automatically.

- **Inflow to this Segment [L^3T^{-1}]:** The streamflow entering the segment. Enter -1, if inflow into this segment is the sum of outflow from its tributary segments. If a segment is a diversion, then the specified inflow is subtracted from flow in the main stream. However, if the specified inflow of the diversion is greater than the flow out of the segment from which flow is to be diverted, then no flow is diverted from that segment.

- **Reach (Vertex Properties):**

The data of this group are assigned to one or more vertices of a polyline. These vertex data may be interpolated to model cells following the rules described in Section 3.4.2.2 and the rules below.

- **Streambed Conductance per Unit Length [LT⁻¹]** is interpolated to the reaches along the polyline. For each of those reaches, the interpolated value C_{strl} is multiplied with the length of the reach within the cell to obtain the streambed conductance C_{str} :

$$C_{str} = C_{strl} \cdot L_c \quad (35)$$

where

C_{strl} is the streambed conductance per unit length at a given cell, and L_c is the length of the reach within the cell.

- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the source water concentration of up to 100 species to current vertex. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.
- **The other parameters:** See the description in the cell-by-cell input method above.

- **Structure:** Same as the cell-by-cell input method.

3.9.1.1.16. Streamflow-Routing (SFR2)

The SFR2⁴⁸ Package (Niswonger, et al., 2006) is used to simulate streams in a model and is an alternative to the Stream (STR) Package, with the most important difference being that stream depth is computed at the midpoint of each reach instead of at the beginning of each reach, as

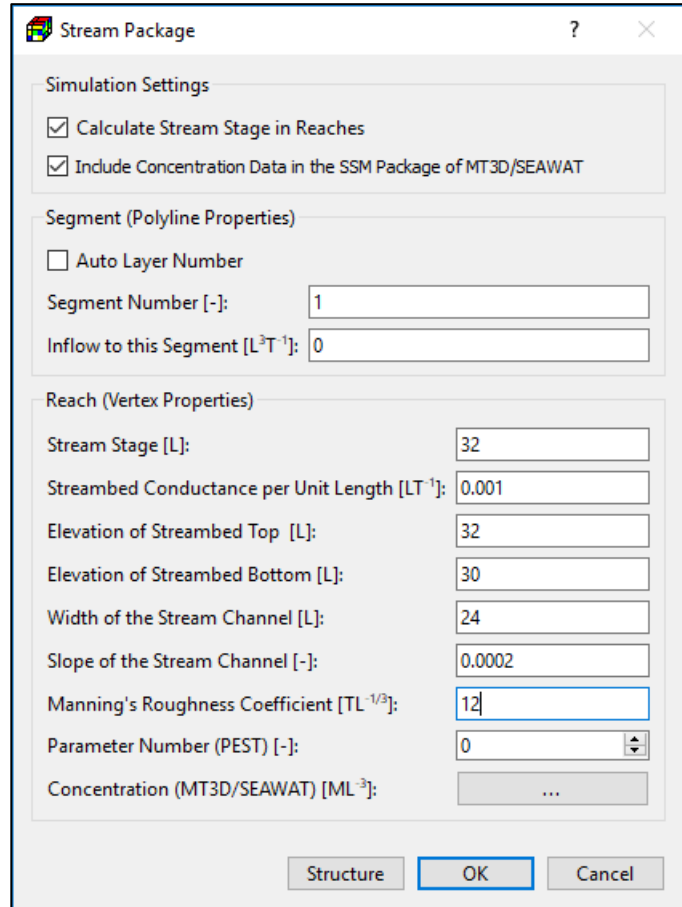


Figure 90. The Stream Package dialog box for specifying vertex properties

⁴⁸ MODFLOW-2005 will not work if the SFR2 Package is activated and the SFR box of Modflow Output Control (Section 3.9.1.3) is checked. If the SFT Package of MT3D-USGS is not used, uncheck the SFR box. If you need to use the SFT Package, the SFR box must be checked and you must set MODFLOW version to either MODFLOW-NWT or MODFLOW-OWHM.

was done in the STR Package. This approach allows for the addition and subtraction of water from runoff, precipitation, and evapotranspiration within each reach. The STR, SFR1⁴⁷, and SFR2 packages should not be used in a model at the same time.

In SFR2, the flow in a stream is either routed instantaneously to downstream streams or routed using a kinematic wave equation. Unsaturated flow is simulated independently of saturated flow within each model cell corresponding to a stream reach whenever the water table (head in MODFLOW) is below the elevation of the streambed. The relation between unsaturated hydraulic conductivity and water content is defined by the Brooks-Corey function. Unsaturated flow variables specified in SFR2 include saturated and initial water contents; saturated vertical hydraulic conductivity; and the Brooks-Corey exponent. These variables are defined independently for each stream reach.

Like the STR Package, the SFR2 Package divides streams into segments and reaches. A segment consists of a group of reaches connected in downstream order. One or more reaches may be assigned to a model cell. The cell-by-cell or polyline input methods may be used to specify the stream data. The polygon input method is not supported. The polyline input method is recommended as it can assign the reach numbers in the required sequential order automatically. The required input parameters are listed below.

1. **All input methods:** The data-entry dialog boxes of all input methods contain an Options tab (Figure 91) with the following common data.
 - **ISFROPT** controls whether vertical unsaturated flow beneath stream reaches is simulated. The following available options are available.
 - *0: No vertical unsaturated flow beneath streams. Time-varying streambed characteristics:* If this option is selected, streambed characteristics are specified for each stress period using the *SFR2 Segments* dialog box (Figure 93).
 - *1: No vertical unsaturated flow beneath streams. Constant streambed characteristics:* If this option is selected, streambed characteristics are assumed to be constant over time and are specified in the *Time-Independent Reach Properties* group.
 - *2: Simulate vertical unsaturated flow beneath streams. Set UHC to VK. Constant streambed characteristics:* If this option is selected, streambed characteristics are assumed to be constant over time and are specified in the *Time-Independent Reach Properties* group. Saturated vertical hydraulic conductivity for the unsaturated zone is the same as the vertical hydraulic conductivity (VK) of the corresponding layer. This option may not be used in conjunction with the BCF Package.
 - *3: Simulate vertical unsaturated flow beneath streams. Use specified UHC. Constant streambed characteristics.* If this option is selected, streambed characteristics are assumed to be constant over time and are specified in the *Time-Independent Reach Properties* group. The saturated vertical hydraulic

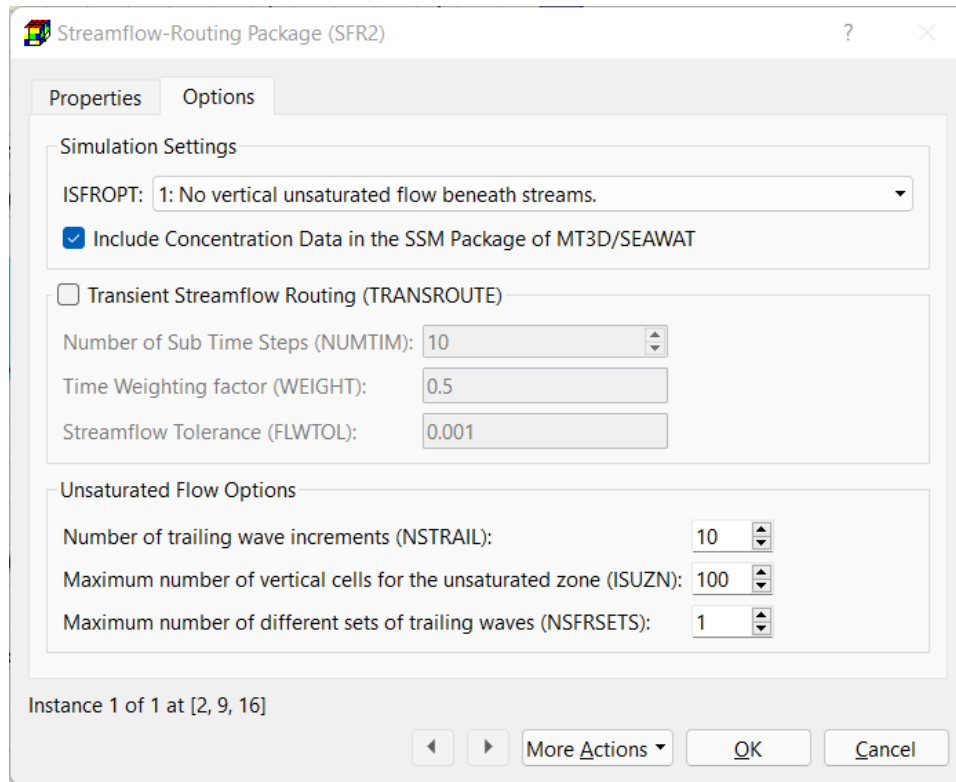


Figure 91. The Options tab of the Streamflow-Routing Package (SFR2) dialog box

- conductivity for the unsaturated zone must be specified in the UHC box in the Time-Independent Reach Properties group.
- 4: *Simulate vertical unsaturated flow beneath streams. Set UHC to VK. Time-varying streamed characteristics.* If this option is selected, streamed characteristics are specified for each stress period using the *SFR2 Segments* dialog box (Figure 93). Saturated vertical hydraulic conductivity for the unsaturated zone is the same as the vertical hydraulic conductivity (VK) of the corresponding layer. This option may not be used in conjunction with the BCF Package.
 - 5: *Simulate vertical unsaturated flow beneath streams. Use specified UHC. Time-varying streamed characteristics.* If this option is selected, streamed characteristics and the saturated vertical hydraulic conductivity (UHC) for the unsaturated zone are specified for each stress period using the *SFR2 Segments* dialog box (Figure 93).
 - **Include Concentration Data in the SSM Package of MT3D/SEAWAT:** This determines how the entered concentration data are used, See *Concentration (MT3D/SEAWAT) [ML⁻³]* below for details.
 - **Transient Streamflow Routing (TRANSROUTE):** When this box is checked, streamflow will be routed using the kinematic-wave equation, see p. 68-69 of (Markstrom, et al., 2008) for details about the kinematic-wave approximation. The following parameters are used to control the behavior of transient streamflow routing:

- **Number of Sub Time Steps (NUMTIM):** NUMTIM is the number of sub time steps used to route streamflow. The time step that will be used to route streamflow will be equal to the MODFLOW time step divided by NUMTIM.
- **Time Weighting factor (WEIGHT):** This is used to calculate the change in channel storage. WEIGHT should have a value between 0.5 and 1. See equation 83 in (Markstrom, et al., 2008) for further details.
- **Streamflow Tolerance (FLWTOL):** This is used for convergence of the kinematic wave equation. A value of 0.00003 m³/s has been used successfully in test simulations of the SFR2 Package.
- **Unsaturated Flow Options:** This group is activated only if ISFROPT is greater than 1.
 - **Number of trailing wave increments (NSTRAIL):** This is the number of trailing wave increments used to represent a trailing wave. Trailing waves are used to represent a decrease in the surface infiltration rate. The value can be increased to improve mass balance in the unsaturated zone. As indicated by (Niswonger, et al., 2006), a value between 10 and 20 works well and results in unsaturated-zone mass balance errors beneath streams ranging between 0.001 and 0.01 percent.
 - **Maximum number of vertical cells for the unsaturated zone (ISUZN):** This is the maximum number of vertical cells used to define the unsaturated zone beneath a stream reach. If ICALC is 1 for all segments then ISUZN should be set to 1 as suggested by (Niswonger, et al., 2006).
 - **Maximum number of different sets of trailing waves (NSFRSETS):** This is the maximum number of different sets of trailing waves used to allocate arrays. Arrays are allocated by multiplying NSTRAIL by NSFRSETS. A value of 30 is usually sufficient for problems where the stream depth varies often. However, in some cases a value of 50 may be required. If this value is set too small, then the simulation will terminate prematurely, and an error will be printed to the Run Listing file.
- 2. **Cell-by-cell input method:** Figure 92 shows the Properties tab of the Streamflow-Routing Package (SFR2) dialog box. The required data are organized in two groups as described below.
 - **Time-Independent Reach Properties:** The following parameters remain unchanged for the entire simulation.
 - **Segment Number (ISEG) [-]:** The number of the segment to which the current cell belongs. Enter 0 to remove the cell from the segment.
 - **Reach Number (IREACH) [-]:** Each model cell in a segment is a reach. IREACH is a sequential number in a segment that begins with one for the farthest upstream reach and continues in downstream order to the last reach in the segment.
 - **Reach Length (RCHLEN) [L]:** Length of the stream reach within the model cell.

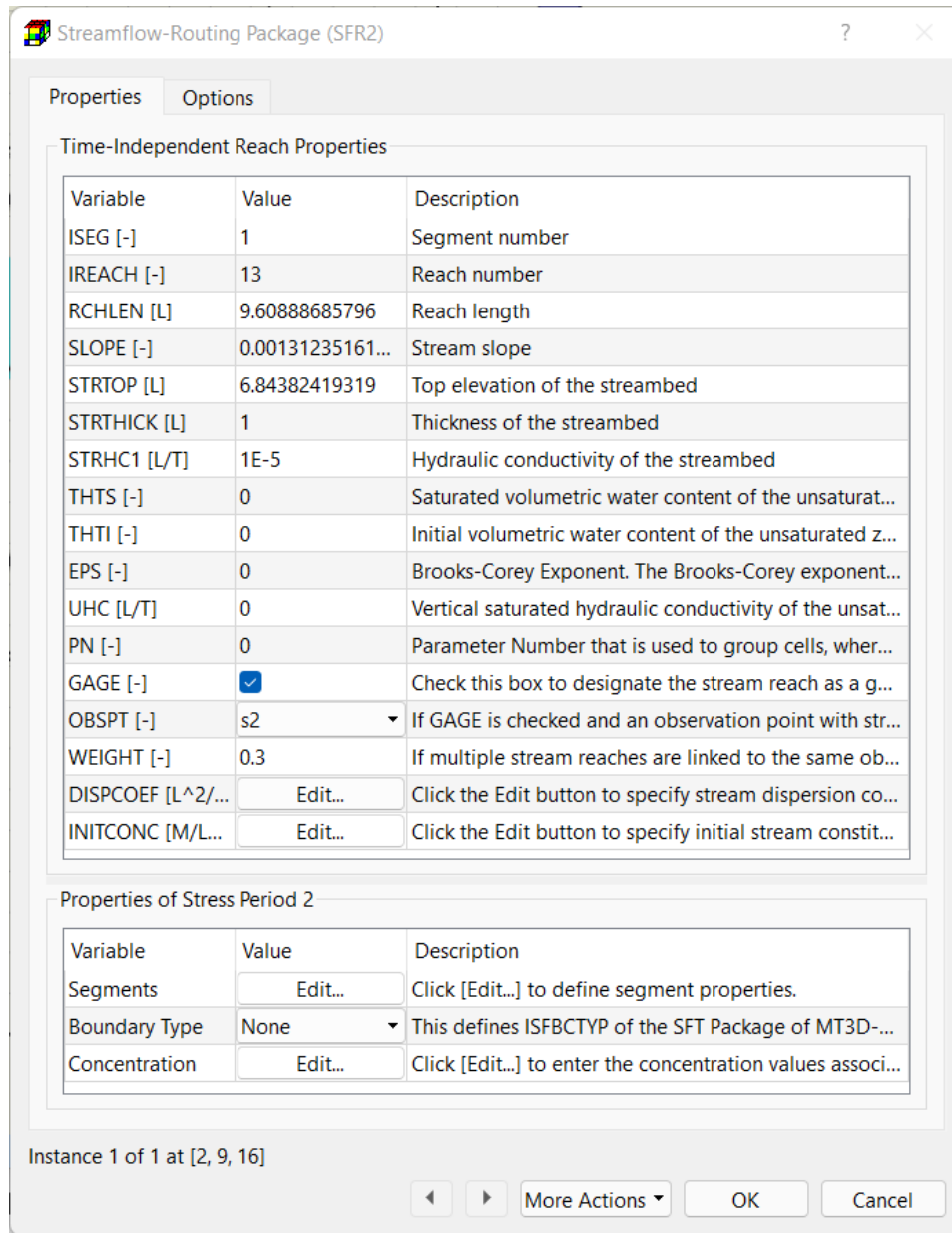


Figure 92. The Streamflow-Routing Package (SFR2) dialog box for specifying cell properties

- **Stream Slope (SLOPE) [-]**: Stream slope across the reach.
- **Top Elevation of the Streambed (STRTOP) [L]**: Average elevation of the top of the streambed within model cell.
- **Thickness of the Streambed (STRTHICK) [L]**: Average thickness of the streambed within the model cell.
- **Streambed Hydraulic Conductivity (STRHC1) [LT⁻¹]**: Average hydraulic conductivity of the streambed within the model cell.
- **Saturated Volumetric Water Content (THTS) [-]**: Value of the saturated volumetric water content in the unsaturated zone beneath the stream reach.

- **Initial Volumetric Water Content (THTI) [-]:** Initial value of the volumetric water content in the unsaturated zone beneath the stream reach. The value of THTI must meet this requirement: $THTS - S_y \leq THTI \leq THTS$.
- **Brooks-Corey Exponent (EPS) [-]:** The Brooks-Corey exponent is used in the relation between water content and hydraulic conductivity within the unsaturated zone.
- **Vertical Saturated Hydraulic Conductivity (UHC) [LT⁻¹]:** Vertical saturated hydraulic conductivity of the unsaturated zone beneath the stream reach. *UHC* is activated only if ISFROPT is set to 3.
- **Parameter Number (PEST) [-]** is used to group cells, where the STRHC1 is to be estimated by PEST.
- **GAGE [-]:** If this box is checked, the simulated stream stage, outflow, as well as depth, width and flow at the midpoint of the reach will be written by the Gage Package to its output file (modflow.streamgageoutput). In addition, the Streamflow Transport (SFT) package of MT3D-USGS will write the simulated concentration in the stream cell to the Streamflow Transport Package Observation file (mt3d.sft.obs).
- **OBSPT [-]:** This dropdown box contains a list of observation points that have at least one streamflow observation data. If *GAGE* is checked and an observation point is selected, the current stream reach is associated with the selected observation point. In that case, the calculated streamflow values at the reach and the observed values at the observation point can be displayed in the Scatter Chart (Section 3.9.1.6.3) and Line Chart (Section 3.9.1.6.4) dialog boxes; in addition, PEST can use the calculated and observed streamflow values for model calibration. See *STREAMFLOWOPT* of the PEST Control Data in Section 3.9.5.2 for available options.
- **WEIGHT [-]:** If multiple stream reaches are linked to the same observation point, the weighted average of the calculated streamflow values in those reaches are used to compare with the observed streamflow by PEST for model calibration.
- **Dispersion Coefficient (MT3D-USGS) [L²T⁻¹]:** Click the Edit button to specify stream dispersion coefficient associated with the current reach. The stream dispersion coefficient can vary for each species. The specified values are used by the Streamflow Transport (SFT) package of MT3D-USGS.
- **Initial Concentration (MT3D-USGS) [ML⁻³]:** Click the Edit button to specify initial stream constituent concentration associated with the current reach. The specified values are used by the Streamflow Transport (SFT) package of MT3D-USGS.
- **Properties for Current Stress Period:** The following properties will be used for the current stress period.
 - **Boundary Type (MT3D-USGS):** This defines the boundary condition (ISFBCTYP) of the current reach in the SFT Package.
 - **Headwater boundary.** That is for streams entering at the boundary of the simulated domain that need a specified concentration

- **Precipitation boundary.** Use this option if precipitation (i.e., PPTSW in Figure 93) directly to channels is simulated in the flow model and a non-zero concentration (default is zero) is desired.
- **Runoff boundary.** Use this option if you want to assign a non-zero concentration (default is zero) associated with the runoff values (i.e., RUNOFF in Figure 93).
- **Constant-concentration boundary.** Any reach number may be set equal to a constant concentration boundary condition.
- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the concentration of up to 100 species to the current reach. The specified concentration data will be used in the following manner:
 - If MT3D-USGS is the selected Transport Model (Figure 116) and its SFT Package is activated, the concentration data will be associated with the specified Boundary Type in the SFT Package. The concentration data will not be used in the SSM Package.
 - If the selected Transport Model is MT3DMS or SEAWAT, and the *Include Concentration Data in the SSM Package of MT3D/SEAWAT* box is checked, the concentration data will be used as specified concentration of the stream water in the SSM Package.
 - If neither of the cases applies, the concentration data will not be used.
- **Segments:** Click the Edit button to open the SFR2 Segments dialog box (Figure 93). This dialog box contains the Segment Table and several buttons as described below. Each row of the table in this dialog box represents a segment of the current stress period. If the value of a cell in the table is not used for the simulation, the cell is grayed out.
 - **Segment Table:** A cell in the Segment Table is grayed out if its data will not be used. The columns of the Segment Table are described below.
 - **NSEG:** Segment number
 - **ICALC** is a flag that controls the calculation of stream depth:
 - **ICALC = 0:** Stream depth is specified by DEPTH1 and DEPTH2 at the beginning of a stress period and remains unchanged unless flow at the midpoint of a reach is zero, then depth is set to zero in that reach. No unsaturated flow is allowed.

NSEG	ICALC	OUTSEG	IUPSEG	IPRIOR	FLOW	RUNOFF	ETSW	PPTSW	ROUGHCH	ROUGHBK	HCOND1	HCOND2	THICKM1	THICKM2	ELEVUP	ELEVDN	WIDTH1	WIDTH2	DEPTH1	DEPTH2
1	2	2	0	0	2168161	0	0	0	3.115497...	3.115497...	0	0	0	0	0	0	0	0	0	0
2	2	4	0	0	2281220	0	0	0	3.115497...	3.115497...	0	0	0	0	0	0	0	0	0	0
3	1	4	0	0	0	0	0	0	3.115497...	0	0	0	0	0	0	0	27	27	0	0
4	2	6	0	0	0	0	0	0	3.115497...	3.115497...	0	0	0	0	0	0	0	0	0	0
5	1	6	0	0	0	0	0	0	3.115497...	0	0	0	0	0	0	0	36	36	0	0

Figure 93. The SFR2 Segments dialog box

- ICALC = 1: Stream depth is calculated using Manning's equation and assuming a wide rectangular channel. Unsaturated flow is simulated when ISFROPT > 1 and the value of ICALC must remain 1 for the specified segment during the simulation.
 - ICALC = 2: Stream depth is calculated using Manning's equation and assuming an eight-point channel cross section for each segment (which allows for the computation of a wetted perimeter and for changing hydraulic conductance of the streambed in relation to changes in flow). Unsaturated flow is simulated when ISFROPT > 1 and the value of ICALC must remain 2 for the specified segment during the simulation.
 - ICALC = 3: Stream depth and width are calculated using equations relating each to streamflow (Q): $DEPTH = CDPTH \times Q^{FDPTH}$ and $WIDTH = AWDTH \times Q^{BWDTH}$. Unsaturated flow is not allowed.
 - ICALC = 4: Stream depth and width are calculated using a table relating streamflow to depth and width. The table is defined by selecting *More Actions > Streamflow Rating Curve* above. Unsaturated flow is not allowed.
- **OUTSEG** is the segment number of the downstream segment that receives tributary inflow from the last reach of this segment. If the segment ends within the modeled grid and OUTSEG = 0, outflow from the segment is not routed anywhere and is no longer part of the stream network. One may wish to use this if all flow in the stream gets diverted into a lined canal or into a pipe. If the flow out of this segment discharges into a lake, set OUTSEG equal to the negative value of the lake identification number.
- **IUPSEG** is the number of the upstream segment from which water is diverted (or withdrawn) to supply inflow to this segment. If the source of a stream segment is discharge from a lake, set IUPSEG equal to the negative value of the lake identification number. If this stream segment does not receive inflow from a lake or as a diversion from an upstream segment, set IUPSEG to zero.
- **IPRIOR** defines the prioritization system for diversion and is used in conjunction with the value of FLOW (see below). IPRIOR is used only if IUPSEG > 0.
- **FLOW** [L^3/T]: If the segment is a head water stream, FLOW is the streamflow entering the first reach of the segment. If the segment is a tributary stream, FLOW is the flow added to or removed from the first reach of the segment. If the segment is a diversionary stream (i.e., IUPSEG is not 0), the following equations are used to define the flow (Qd) that may be diverted to this segment from the IUPSEG segment that has the available flow of Q.
 - IPRIOR = 0: $Qd = \min(FLOW, Q)$
 - IPRIOR = -1: $Qd = 0$ if $FLOW > Q$ otherwise $Qd = FLOW$.
 - IPRIOR = -2: $Qd = FLOW \times Q$, in this case $0.0 \leq FLOW \leq 1.0$
 - IPRIOR = -3: $Qd = Q$ if $FLOW > Q$ otherwise $Qd = 0$

If *tabular inflow* data (see *More Actions* below) of a segment is present, inflow to the segment is specified by the tabular inflow data and *FLOW* is ignored.

- **RUNOFF** [L³/T] is the volumetric rate of the diffuse overland runoff that enters the stream segment.
- **ETSW** [L/T] is the volumetric rate per unit area of water removed by evapotranspiration directly from the stream channel. ETSW is defined as a positive value.
- **PPTSW** [L/T] is the volumetric rate per unit area of water added by precipitation directly on the stream channel.
- **ROUGHCH** is Manning's roughness coefficient for all reaches in this segment. See Section 3.9.1.1.15 for a discussion of Manning's roughness coefficient.
- **ROUGHBK** is Manning's roughness coefficient for the overbank area in all reaches in this segment. This value is used only if ICALC = 2.
- **CDPTH** and **FDPTH**: These are the coefficients used in the equation: (DEPTH = CDPTH × Q^{FDPTH}) that relates stream depth in all reaches in this segment to streamflow. These variables are only used if ICALC = 3.
- **AWDTH** and **BWDTH**: These are the coefficients used in the equation: (WIDTH = AWDTH × Q^{BWDTH}) that relates stream width in all reaches in this segment to streamflow. These variables are only used if ICALC = 3.
- **HCOND1** [L] and **HCOND2** [L] are the hydraulic conductivity values of the streambed at the upstream end and downstream end of this segment, respectively.
- **THICKM1** [L] and **THICKM2** [L] are the thicknesses of streambed material at the upstream end and downstream end of this segment, respectively.
- **ELEVUP** [L] and **ELEVDN** [L] are the elevations of the top of the streambed at the upstream end and downstream end of this segment, respectively.
- **WIDTH1** [L] and **WIDTH2** [L] are the widths of the stream channel at the upstream end and downstream end of this segment, respectively.
- **DEPTH1** [L] and **DEPTH2** [L] are the depths of water in the stream channel at the upstream end and downstream end of this segment, respectively.
- **THTS1** [-] and **THTS2** [-] are the saturated volumetric water content in the unsaturated zone beneath the upstream end and downstream end of this segment, respectively.
- **THTI1** [-] and **THTI2** [-] are the initial volumetric water content in the unsaturated zone beneath the upstream end and downstream end of this segment, respectively. THTI must be less than or equal to THTS and greater than or equal to THTS minus the specific yield.
- **EPS1** [-] and **EPS2** [-] are the Brooks-Corey exponents used in the relation between water content and hydraulic conductivity within the unsaturated zone beneath the upstream end and downstream end of this segment, respectively.

- **UHC1** [L/T] and **UHC2** [L/T] are the values of the vertical saturated hydraulic conductivity of the unsaturated zone beneath the upstream end and downstream end of this segment, respectively.
- **Add:** Click this button to add a row to the table.
- **Delete:** Click this button to delete the selected row(s) from the table.
- **More Actions:**
 - **X-Section:** Select this item to open the *Eight-point Cross Section* dialog box that allows you to edit the 8-point channel cross-section of the selected row. If more than one rows are selected, the data of the row with the smallest segment number will be edited. The *Eight-point Cross Section* dialog box contains a table with two columns – XCPT and ZCPT. XCPT is the distance relative to the left bank of the stream channel (when looking downstream) for the eight points used to describe the geometry of the segment of the stream channel. ZCPT is the height relative to the top of the lowest elevation of the streambed. The lowest elevation (ZCPT = 0.0) can be any one from Point 2 through 7. The part between points 1 and 3 represents the left bank; the part between points 3 and 6 represents the channel; and the part between points 6 and 8 represents the right bank. Different Manning's roughness coefficients (see ROUGHCH and ROUGHBK below) may be assigned the channel and the left and right overbank areas.
 - **Streamflow Rating Curve:** Select this action to open the Streamflow Rating dialog box that allows you to edit the following values of the segment. These values are used only if ICALC = 4 (see below).
 - **FLOWTAB:** The streamflow [L³/T] related to a given depth and width. One value is needed for each streamflow that has a corresponding value of depth and width.
 - **DPTH TAB:** This is the average stream depth [L] corresponding to a given streamflow.
 - **WDHTAB:** This is the average stream width [L] corresponding to a given streamflow.
 - **Tabular Inflow:** Select this item to open the Time Series dialog box⁴⁹ (Figure 62) where you can edit an inflow time series for the segment NSEG of the selected row. The Time Series dialog box contains a table with two columns: Time and Value. Time is the simulation time when the inflow is specified for the segment; Value is the specified flow [L³/T]. Times listed in the table do not need to correspond to the beginning of time steps. If the beginning of a time step falls between times listed in the tabular flow data, then the specified inflow is calculated by SFR2 using a time-weighted average of specified flows within the time step. Times can be listed in the table either more frequently or less frequently than the time steps. When tabular inflow data of a segment is

⁴⁹ See Section 3.9.1.1.5 for details of the Time Series dialog box.

- present, it replaces the data entered to the FLOW column in the Segment Table. The Tabular Inflow data is specified for the entire simulation.
- **Import Tabular Inflow:** Select this item to import inflow data from one or more Tabular Inflow files (Section 5.1.16). The existing inflow data of stream segments are retained if they are not included in the imported file(s).
 - **Clear Tabular Inflow:** Select this item to delete tabular inflow data of all segments.
 - **Export:** Select this item to export the complete set of segment data to a Segment data file.
 - **Import:** Select this item to import segment data from a Segment data file.
- **Table:** A cell in the table is grayed out if its data will not be used. The columns of the table are described below.
- **NSEG:** Segment number
 - **ICALC** is a flag that controls the calculation of stream depth:
 - ICALC = 0: Stream depth is specified by DEPTH1 and DEPTH2 at the beginning of a stress period and remains unchanged unless flow at the midpoint of a reach is zero, then depth is set to zero in that reach. No unsaturated flow is allowed.
 - ICALC = 1: Stream depth is calculated using Manning's equation and assuming a wide rectangular channel. Unsaturated flow is simulated when ISFROPT > 1 and the value of ICALC must remain 1 for the specified segment during the simulation.
 - ICALC = 2: Stream depth is calculated using Manning's equation and assuming an eight-point channel cross section for each segment (which allows for the computation of a wetted perimeter and for changing hydraulic conductance of the streambed in relation to changes in flow). Unsaturated flow is simulated when ISFROPT > 1 and the value of ICALC must remain 2 for the specified segment during the simulation.
 - ICALC = 3: Stream depth and width are calculated using equations relating each to streamflow (Q): $DEPTH = CDPTH \times Q^{FDPTH}$ and $WIDTH = AWDTH \times Q^{BWDTH}$. Unsaturated flow is not allowed.
 - ICALC = 4: Stream depth and width are calculated using a table relating streamflow to depth and width. The table is defined by selecting *More Actions > Streamflow Rating Curve* above. Unsaturated flow is not allowed.
 - **OUTSEG** is the segment number of the downstream segment that receives tributary inflow from the last reach of this segment. If the segment ends within the modeled grid and OUTSEG = 0, outflow from the segment is not routed anywhere and is no longer part of the stream network. One may wish to use this if all flow in the stream gets diverted into a lined canal or into a pipe. If the flow out of this segment discharges into a lake, set OUTSEG equal to the negative value of the lake identification number.

- **IUPSEG** is the number of the upstream segment from which water is diverted (or withdrawn) to supply inflow to this segment. If the source of a stream segment is discharge from a lake, set IUPSEG equal to the negative value of the lake identification number. If this stream segment does not receive inflow from a lake or as a diversion from an upstream segment, set IUPSEG to zero.
- **IPRIOR** defines the prioritization system for diversion and is used in conjunction with the value of FLOW (see below). IPRIOR is used only if IUPSEG > 0.
- **FLOW** [L³/T]: If the segment is a head water stream, FLOW is the streamflow entering the first reach of the segment. If the segment is a tributary stream, FLOW is the flow added to or removed from the first reach of the segment. If the segment is a diversionary stream (i.e., IUPSEG is not 0), the following equations are used to define the flow (Qd) that may be diverted to this segment from the IUPSEG segment that has the available flow of Q.
 - IPRIOR = 0: Qd = min(FLOW, Q)
 - IPRIOR = -1: Qd = 0 if FLOW > Q otherwise Qd = FLOW.
 - IPRIOR = -2: Qd = FLOW x Q, in this case 0.0 ≤ FLOW ≤ 1.0
 - IPRIOR = -3: Qd = Q if FLOW > Q otherwise Qd = 0
- **RUNOFF** [L³/T] is the volumetric rate of the diffuse overland runoff that enters the stream segment.
- **ETSW** [L/T] is the volumetric rate per unit area of water removed by evapotranspiration directly from the stream channel. ETSW is defined as a positive value.
- **PPTSW** [L/T] is the volumetric rate per unit area of water added by precipitation directly on the stream channel.
- **ROUGHCH** is Manning's roughness coefficient for all reaches in this segment. See Section 3.9.1.1.15 for a discussion of Manning's roughness coefficient.
- **ROUGHBK** is Manning's roughness coefficient for the overbank area in all reaches in this segment. This value is used only if ICALC = 2.
- **CDPTH** and **FDPTH**: These are the coefficients used in the equation: (DEPTH = CDPTH × Q^{FDPTH}) that relates stream depth in all reaches in this segment to streamflow. These variables are only used if ICALC = 3.
- **AWDTH** and **BWDTH**: These are the coefficients used in the equation: (WIDTH = AWDTH × Q^{BWDTH}) that relates stream width in all reaches in this segment to streamflow. These variables are only used if ICALC = 3.
- **HCOND1** [L] and **HCOND2** [L] are the hydraulic conductivity values of the streambed at the upstream end and downstream end of this segment, respectively.
- **THICKM1** [L] and **THICKM2** [L] are the thicknesses of streambed material at the upstream end and downstream end of this segment, respectively.

- **ELEVUP** [L] and **ELEVDN** [L] are the elevations of the top of the streambed at the upstream end and downstream end of this segment, respectively.
- **WIDTH1** [L] and **WIDTH2** [L] are the widths of the stream channel at the upstream end and downstream end of this segment, respectively.
- **DEPTH1** [L] and **DEPTH2** [L] are the depths of water in the stream channel at the upstream end and downstream end of this segment, respectively.
- **THTS1** [-] and **THTS2** [-] are the saturated volumetric water content in the unsaturated zone beneath the upstream end and downstream end of this segment, respectively.
- **THTI1** [-] and **THTI2** [-] are the initial volumetric water content in the unsaturated zone beneath the upstream end and downstream end of this segment, respectively. THTI must be less than or equal to THTS and greater than or equal to THTS minus the specific yield.
- **EPS1** [-] and **EPS2** [-] are the Brooks-Corey exponents used in the relation between water content and hydraulic conductivity within the unsaturated zone beneath the upstream end and downstream end of this segment, respectively.
- **UHC1** [L/T] and **UHC2** [L/T] are the values of the vertical saturated hydraulic conductivity of the unsaturated zone beneath the upstream end and downstream end of this segment, respectively.

3. Polyline input method: Figure 94 shows the Properties tab of the Streamflow-Routing Package (SFR2) dialog box. The required data are organized in two groups as described below. Once the vertex data are specified, they may be interpolated and applied to model cells following the rules described in Section 3.4.2.2. Note that the specified values of GAGE, OBSPT, and WEIGHT only apply to the stream reach at the vertex location and will not be interpolated to reaches between vertices.

- **Time-Independent Reach Properties**

- **Layer Number**

- **Current Layer:** Assign reaches along the polyline to the current layer.
- **Highest Active Layer:** Assign the reaches along the polyline to the highest active layer.
- **Streambed Elevation:** Assign the reaches along the polyline to a layer where the elevation of the streambed bottom (STRTOP - STRTHICK) lies between the layer's upper and lower bounds. A reach is set to the highest/lowest layer, if (STRTOP - STRTHICK) is higher/lower than the model top/bottom.
- **Highest Active Layer & Streambed Elevation:** Assign the reaches along the polyline to the highest active layer where the elevation of the streambed bottom lies between the layer's upper and lower bounds. If the streambed bottom of a reach does not lie within a layer's lower and upper bounds, the reach is assigned to the highest active layer.

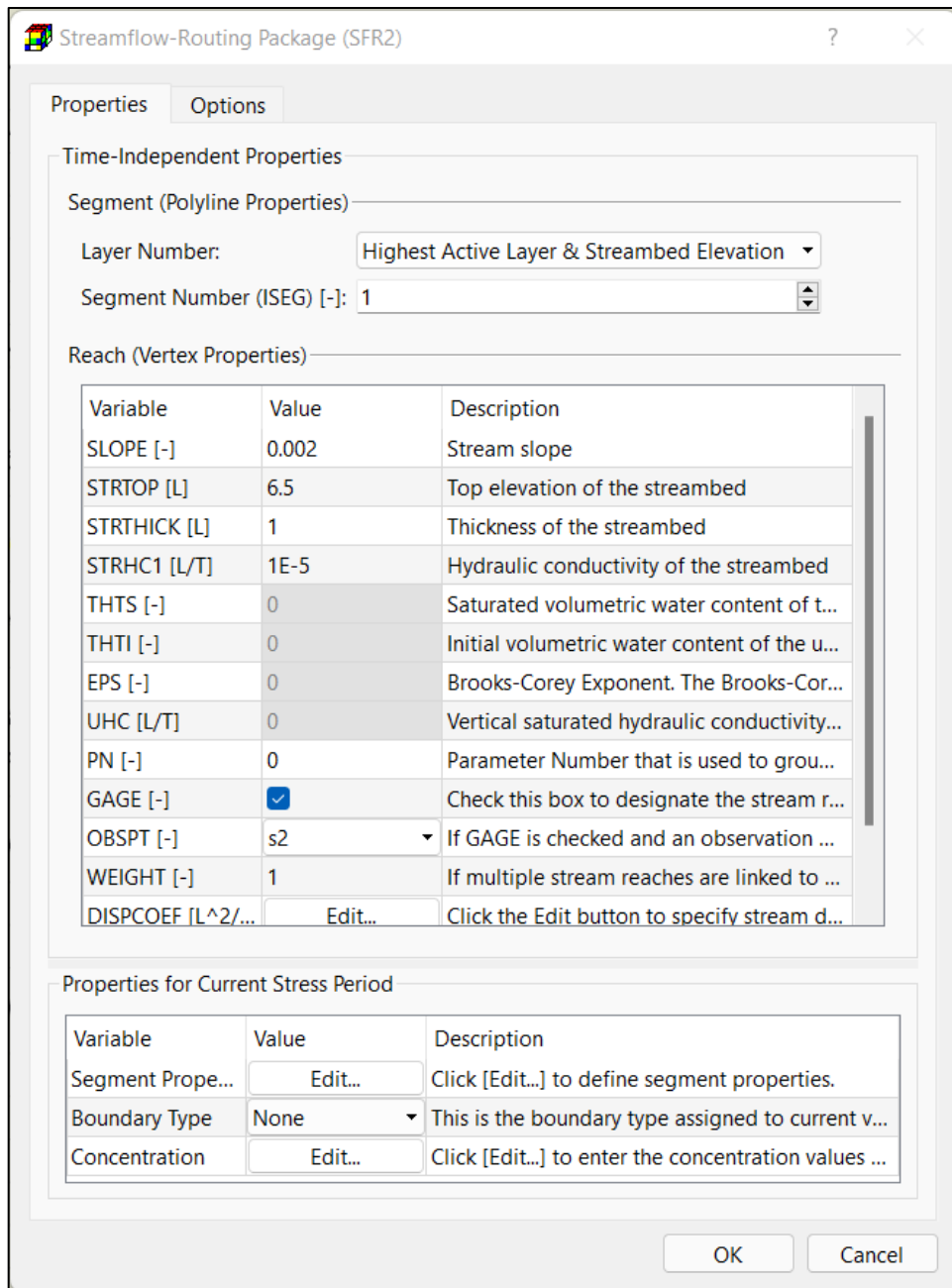


Figure 94. The Streamflow-Routing Package (SFR2) dialog box for specifying vertex properties

- **Segment Number (ISEG) [-]:** The number of the segment to which the reaches along the polyline belong. Prior to applying a polyline to model cells, all existing reaches of the given segment number will be removed. Reach numbers will be assigned to the reaches along the polyline automatically.
- **Reach (Vertex Properties):** See cell-by-cell input method above for the description of the parameters in this group.
- **Properties for Current Stress Period:** The following properties will be used for the current stress period.

- **Concentration (MT3D/SEAWAT) [ML⁻³]:** Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the concentration of up to 100 species to the current vertex. Depending on the selected Concentration option in the Option tab, the entered values may be used as the source water concentration in the SSM package or the specified concentration of the selected boundary type in the SFT package.
- **Segments:** Same as the cell-by-cell input method above.

3.9.1.1.17. Subsidence and Aquifer-System Compaction (SUB)

The SUB Package (Hoffmann, et al., 2003) simulates elastic (recoverable) compaction and expansion, and inelastic (permanent) compaction of compressible fine-grained beds (interbeds) within the aquifers. The deformation of the interbeds is caused by head or pore-pressure changes, and thus by changes in effective stress within the interbeds. If the stress is less than the preconsolidation stress of the sediments, the deformation is elastic; if the stress is greater than the preconsolidation stress, the deformation is inelastic. The propagation of head changes within the interbeds is defined by a transient, one-dimensional (vertical) diffusion equation. This equation accounts for delayed release of water from storage or uptake of water into storage in the interbeds. Properties that control the timing of the storage changes are vertical hydraulic diffusivity and interbed thickness. The SUB Package supersedes the IBS Package (Section 3.9.1.1.8), which assumes that water is released from or taken into storage with changes in head in the aquifer within a single model time step and, therefore, can only be reasonably used to simulate thin interbeds. The SUB Package relaxes this assumption and can be used to simulate time-dependent drainage and compaction of thick interbeds and confining units. The time-dependent drainage can be turned off, in which case the SUB Package gives results identical to those from IBS.

The required input parameters for the cell-by-cell input methods are listed below. The polyline and polygon input methods are not supported.

1. **Cell-by-cell input method:** Figure 95 shows the dialog box for specifying cell properties.
 - **Data Tab**
 - **No-delay Interbeds**
 - **Number of Interbeds (NNDB) [-]:** This is the number of the systems of no-delay interbeds.
 - **Layer Assignment (LN):** Click the Edit button to open the Layer Assignment dialog box that allows you to assign systems no-delay interbeds to model layers.
 - **Properties of Interbed Systems:** the following values need to be assigned to each system of no-delay interbeds:
 - **HC [L]:** this is the preconsolidation head. For any model cells in which the specified HC is greater than the starting hydraulic head, the value of the starting head will be used in the simulation instead of HC.
 - **Sfe [-]:** this is the dimensionless elastic skeletal storage coefficient.

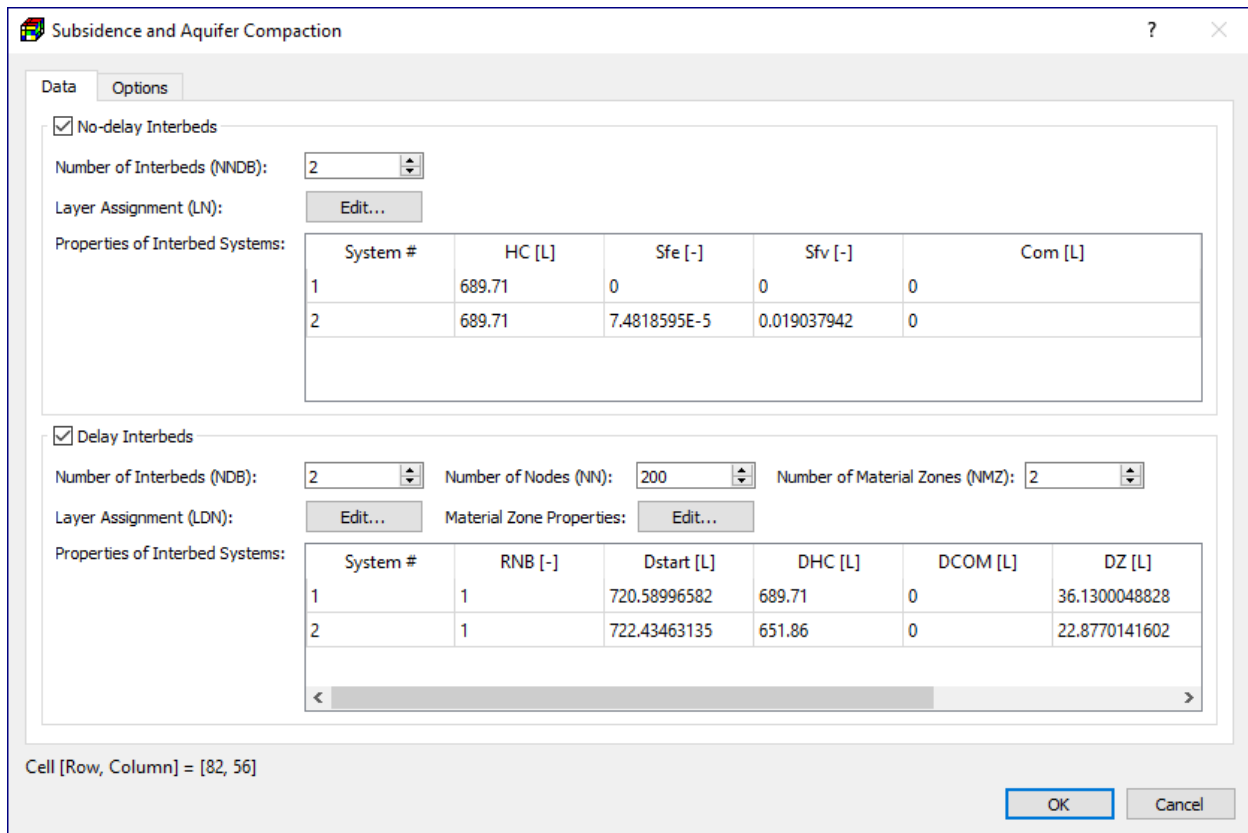


Figure 95. The Subsidence and Aquifer Compaction dialog box

- **Sfv [-]:** this is the dimensionless inelastic skeletal storage coefficient.
 - **Com [L]:** this is the starting compaction. The value of *Com* does not affect the calculation. It is merely added to the computed compaction value so that the stored result values of compaction and subsidence may include previous components defined by *Com*.
- **Delay Interbeds**
 - **Number of Interbeds (NDB):** This is the number of the systems of delay interbeds. To reduce the number of computations required, delay interbeds with the same vertical hydraulic conductivity and elastic and inelastic skeletal specific storage within one model layer can be grouped into one system of delay interbeds.
 - **Number of Nodes (NN):** This is used to discretize the half of the equivalent thickness (*DZ*) of a system of delay interbeds to approximate the head distributions within the system of delay interbeds.
 - **Number of Material Zones (NMZ):** NMZ is the number of material zones that are needed to define the hydraulic properties of systems of delay interbeds. Each material zone is defined by a combination of vertical hydraulic conductivity [LT^{-1}], elastic skeletal specific storage [L^{-1}], and inelastic skeletal specific storage [L^{-1}].

- **Layer Assignment (LDN):** Click the Edit button to open the Layer Assignment dialog box that allows you to assign systems delay interbeds to model layers.
- **Material Zone Properties:** Click the Edit button to open the Material Zone Properties dialog box that allows you to assign vertical hydraulic conductivity [L^{-1}], elastic skeletal specific storage [L^{-1}], and inelastic skeletal specific storage [L^{-1}] to each material zone.
- **Properties of Interbed Systems:** the following values need to be assigned to the model cells of each system of delay interbeds:
 - **RNB [-]** specifies the factor n_{equiv} for the systems of delay interbeds. See eq. 20 of (Hoffmann, et al., 2003) for details. RNB is also used to define the areal extent of each system of interbeds. For cells beyond the areal extent of the system of interbeds, enter a number less than 1.0.
 - **Dstart [L]** is the starting head values in systems of delay interbeds.
 - **DHC [L]** is the starting preconsolidation head for systems of delay interbeds. For any model cells in which the specified DHC is greater than the starting hydraulic head, the value of the starting head will be used in the simulation instead of HC.
 - **DCOM [L]** is the starting compaction in systems of delay interbeds. The value of *DCOM* does not affect the calculation. It is merely added to the computed compaction value so that the stored result values of compaction and subsidence may include previous components defined by *DCOM*.
 - **DZ [L]** is the equivalent thickness of systems of delay interbeds. See eq. 19 of (Hoffmann, et al., 2003) for details.
 - **NZ [-]** is the zone number for systems of delay interbeds. The zone number and the Material Zone Properties define the vertical hydraulic conductivity, elastic skeletal specific storage, and inelastic skeletal specific storage at the cell location.
- **Options Tab (Figure 96)**
 - **Solver**
 - **Acceleration Parameter (AC1):** This is ω_1 in equation 25 of (Hoffmann, et al., 2003). It is used to predict the aquifer head at the interbed boundaries based on the head change computed for the previous iteration. A value of 0.0 results in the use of the aquifer head at the previous iteration. Limited experience indicates that optimum values may range from 0.0 to 0.6.
 - **Acceleration Parameter (AC2):** This is ω_2 in equation 27 of (Hoffmann, et al., 2003). It is a multiplier for the head changes to compute the head at the new iteration. Values are normally between 1.0 and 2.0. This parameter also can be used to help convergence of the iterative solution by using values between 0 and 1.
 - **Minimum Number of Iterations (ITMIN):** This is the minimum number of iterations for which one-dimensional equations will be solved for flow in interbeds when the Strongly Implicit Procedure (SIP) is used to solve the

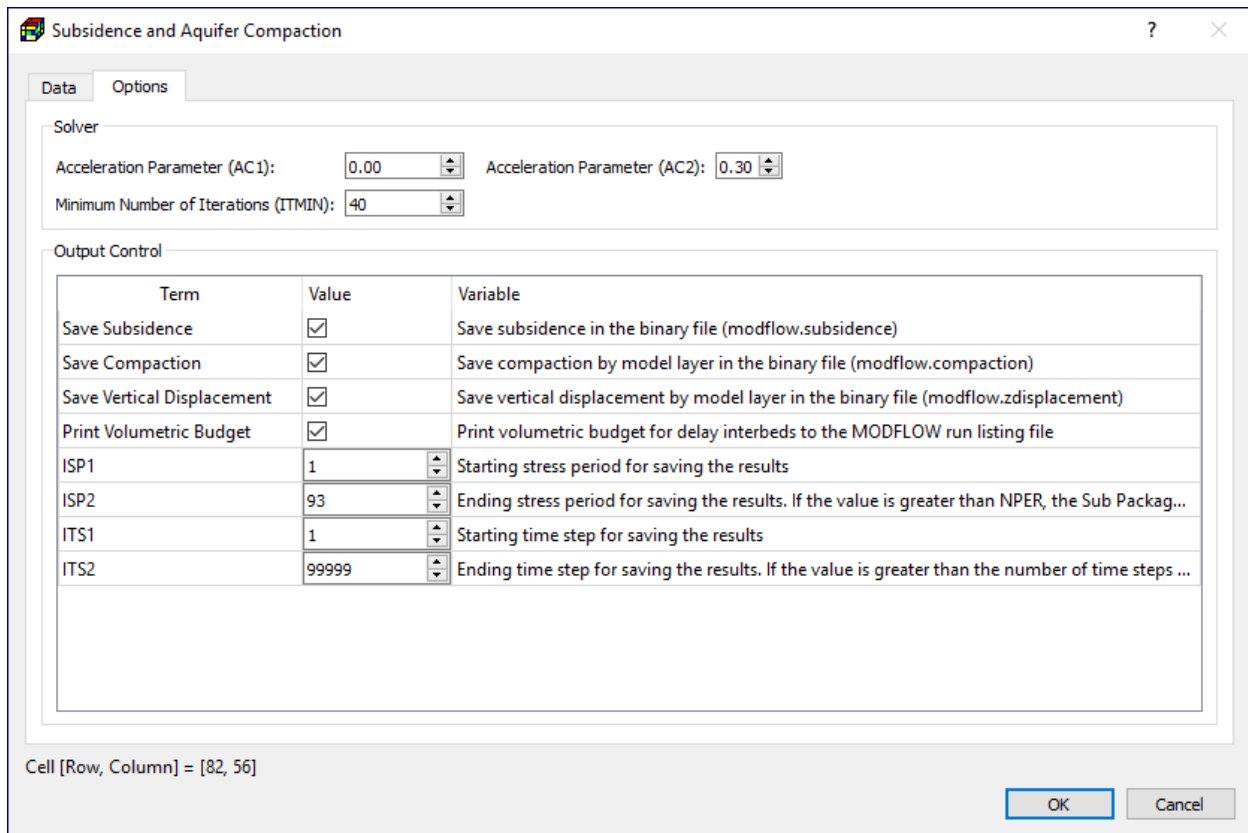


Figure 96. The Options tab of the Subsidence and Aquifer Compaction dialog box

ground-water flow equations. If the current iteration level is greater than ITMIN and the SIP convergence criterion for head closure (HCLOSE) is met at a cell, the one-dimensional equations for that cell will not be solved. The previous solution will be used. The value of ITMIN is not used if a solver other than SIP is used to solve the ground-water flow equations.

o Output Control

- **Save Subsidence:** Check this box to instruct the SUB Package to save subsidence in the binary file modflow.subsidence. Subsidence is the sum of the compaction from all interbed systems, including no-delay and delay systems.
- **Save Compaction:** Check this box to instruct the SUB Package to save compaction by model layer in the binary file modflow.compaction. The SUB Package computes compaction for each system of interbeds. The model layer numbers to which each system belongs are specified in arrays LN and LDN for no-delay and delay systems, respectively. Each model layer can include more than one system of interbeds of either type or combinations of both types. The output option of compaction by model layer is the sum of compaction of all systems within each model layer.
- **Save Vertical Displacement:** Check this box to instruct the SUB Package to save vertical displacement by model layer in the binary file modflow.zdisplacement. Vertical displacement for a model layer is defined as the sum of the compaction

in the layer and in all underlying layers. This displacement corresponds to movement of the upper surface of the model layer. The vertical displacement for layer 1 is equal to the subsidence. Any layers below the lowest system of compressible interbeds will have zero vertical displacement.

- **Print Volumetric Budget:** Check this box to instruct the SUB Package to print volumetric budget for delayed interbeds to the MODFLOW run listing file.
- **ISP1** is the starting stress period for saving the results.
- **ISP2** is the ending stress period for saving the results. If the value is greater than NPER, the Sub Package will change it to NPER.
- **ITS1** is the starting time step for saving the results.
- **ITS2** is the ending time step for saving the results. If the value is greater than the number of time steps in a given stress period, the Sub Package will change it to the number of time steps in that stress period.

3.9.1.1.18. Seawater Intrusion 2 (SWI2)

The SWI2 Package (Bakker, et al., 2013) allows three-dimensional vertically integrated variable-density groundwater flow and seawater intrusion in coastal multi-aquifer systems to be simulated using MODFLOW-2005. The main advantage of the formulation used in the SWI2 Package compared to models that solve flow and transport equations is that the SWI2 Package represents three-dimensional vertically integrated variable-density flow without the need to discretize the aquifer vertically. Vertically integrated variable-density groundwater flow is based on the Dupuit approximation in which an aquifer is vertically discretized into zones of differing densities, separated from each other by defined surfaces representing interfaces or density isosurfaces. The numerical approach used in the SWI2 Package does not account for diffusion and dispersion and should not be used where these processes are important. The resulting differential equations are equivalent in form to the groundwater flow equation for uniform-density flow. The approach implemented in the SWI2 Package allows density effects to be incorporated into MODFLOW-2005 through the addition of pseudo-source terms to the groundwater flow equation without the need to solve a separate advective-dispersive transport equation. Vertical and horizontal movement of defined density surfaces is calculated separately using a combination of fluxes calculated through solution of the groundwater flow equation and a simple tip and toe tracking algorithm.

The required input parameters for all input methods are identical and listed below.

1. All input methods:

- **Data Tab (Figure 97)**
 - **Number of ZETA Surfaces (NSRF) [-]:** This is the number of active surfaces (i.e., interfaces between zones).
 - **Toe Slope (TOESLOPE) [-]:** Maximum slope of toe cells
 - **Tip Slope (TIPSLOPE) [-]:** Maximum slope of tip cells

- **ALPHA [-]:** Fraction of threshold used to move the tip and toe to adjacent empty cells when the slope exceeds user specified TOESLOPE and TIPSLOPE values. ALPHA must be greater than 0.0 and less than or equal to 1.0.

- **BETA [-]:** Fraction of threshold used to move the toe to adjacent non-empty cells when the surface is below a minimum value defined by the

user-specified TOESLOPE value. BETA must be greater than 0.0 and less than or equal to 1.0.

- **Density Distribution Option (ISTRAT) [-]:** This is a flag indicating density distribution.

- **Dimensionless Density (NU) [-]:** The dimensionless density ν is defined as $\nu = (\rho - \rho_f) / \rho_f$, where ρ is the fluid density [M/L³], and ρ_f is the density of freshwater [M/L³]. Click the Edit button to specify the NU values in a Density dialog box.

- If ISTRAT is “density varies linear between surfaces”, NSRF + 1 density values are to be specified. Each value represents the density of the corresponding zone.
- If ISTRAT is “density is constant between surfaces”, NSRF + 2 density values are to be specified. The first value is the density along the top of highest zone (i.e., first zone), followed by density values along each interface, and the density value at the bottom of the lowest zone (i.e. last zone).

- **ISOURCE:** Source type of any external sources or sinks, specified with any outside package (i.e., WEL, RCH, or GHB Package). There are three options:

- If ISOURCE > 0: Sources and sinks have the same fluid density as the zone ISOURCE. If such a zone is not present in the cell, sources and sinks have the same fluid density as the active zone at the top of the aquifer.
- If ISOURCE = 0: Sources and sinks have the same fluid density as the active zone at the top of the aquifer.

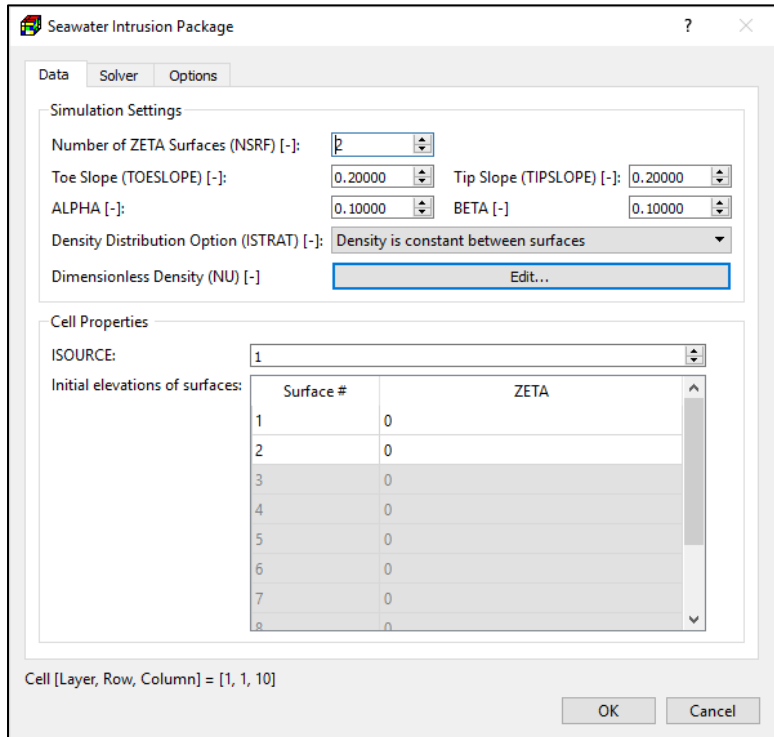


Figure 97. The Data tab of the Seawater Intrusion Package dialog box

- If ISOURCE < 0: Sources have the same fluid density as the zone with a number equal to the absolute value of ISOURCE. Sinks have the same fluid density as the active zone at the top of the aquifer. This option is useful for the modeling of the ocean bottom where infiltrating water is salt, yet exfiltrating water is of the same type as the water at the top of the aquifer.
- **Initial Elevations of Surfaces:** Use the table to specify the initial elevations of the ZETA surfaces (i.e., interface between zones).
 - An elevation for each surface needs to be specified for every cell in the model, even though a zeta surface is two-dimensional in most cases.
 - When the specified Zeta surface value is above the top of the model layer, SWI2 requires the Zeta surface value to be set to the top of model layer in its input file, similarly when the specified Zeta surface value is below the bottom of model layer, SWI2 requires the Zeta surface value to be set to the bottom of model layer. PM will automatically adjust the Zeta surface based on this requirement when creating the input file for SWI2. Therefore, if a surface is present at only one point in the vertical everywhere, the same grid of ZETA values may be entered for every model layer.
 - It is important to start with a reasonably smooth variation of the ZETA surface because it is not physically reasonable for a surface to be discontinuous or have sharp changes in gradient. When an irregular ZETA surface is entered, additional time in a transient simulation may be required before a physically realistic ZETA elevation is achieved in each cell, and it may take small time steps to reach an accurate solution.
- **Solver Tab (Figure 98)**
 - **Solver (NSOLVER) [-]:** The solver used to solve for ZETA surfaces.
 - **Printout Option (MUTSOL):** This controls printing of convergence information from the solver.
 - **Printout Interval (IRPSOL):** The extreme head change and residual (positive or negative) are printed for each iteration of a time step whenever the time step is an even multiple of the printout interval. The printout also occurs at the end of each stress period regardless of the value of the printout interval.
 - **NPCOND:** This is the matrix preconditioning method.
 - **NBPOL:** This is used with the Neuman series polynomial preconditioning method. Checking this box to calculate the estimate of the upper bound on the maximum eigenvalue. If this box is cleared, the estimate is set as 2. Convergence is generally insensitive to this parameter.
 - **Maximum Number of Outer Iterations (MXITER):** For a linear problem MXITER should be 1, unless more than 50 inner iterations are required, when MXITER could be as large as 10. A larger number (generally less than 100) is required for a nonlinear problem.

- **Maximum Number of Inner Iterations (ITERI):**

For nonlinear problems, ITERI usually ranges from 3 to 10; a value of 30 will be sufficient for most linear problems.

- **Relaxation Parameter (RELAX):**

This is the relaxation parameter used with the Modified Incomplete Cholesky preconditioning method. Usually, RELAX=1.0, but for some problems a value of 0.99, 0.98, or 0.97 will reduce the number of iterations required for convergence.

- **Damping Parameters**

- **DAMP**

- If DAMP > 0, it applies to both steady-state and transient stress periods.
 - If DAMP < 0, it applies to steady-state periods. The absolute value is used as the damping factor.

- **DAMPT** is the damping factor for transient stress periods and is only used when DAMP is specified as a negative value.

- **ZETA Change (ZCLOSE) [L]:** This is the ZETA change criterion for convergence. When the maximum absolute value of ZETA change from all nodes during an iteration is less than or equal to ZCLOSE, and the criterion for RCLOSE is also satisfied, iteration stops.

- **Residual (RCLOSE) [L³/T]:** This is the residual criterion for convergence. When the maximum absolute value of the residual at all nodes during an iteration is less than or equal to RCLOSE, and the criterion for ZCLOSE is also satisfied, iteration stops.

- **Options Tab**

- **Adaptive Time Stepping:** Check this box to enable adaptive time stepping algorithm that dynamically adjust the number of SWI2 time steps in a MODFLOW time step. The algorithm has no effect on the lengths of MODFLOW time steps and

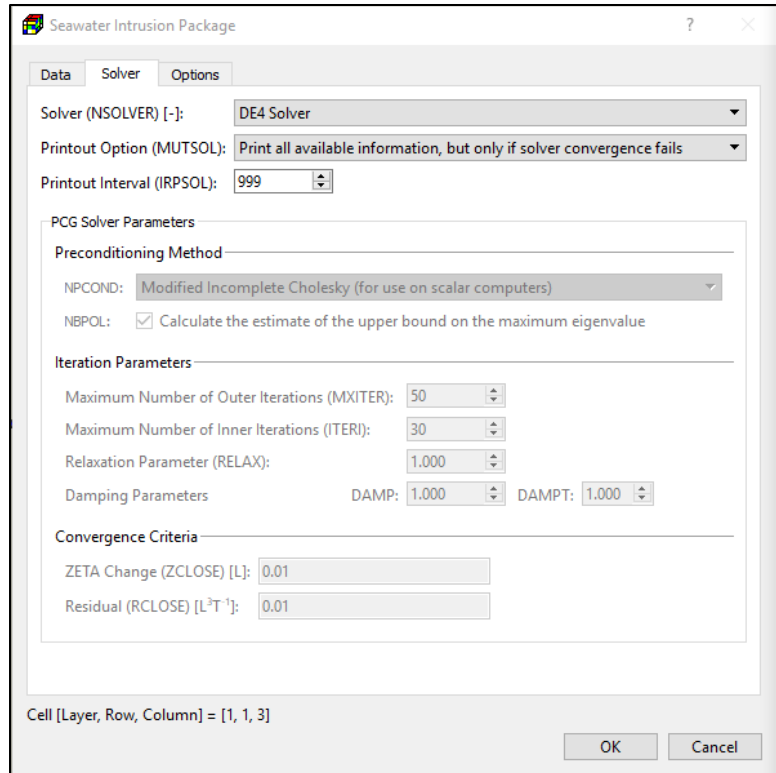


Figure 98. The Solver tab of the Seawater Intrusion Package dialog box

stress periods. Use of this option will increase the number of SWI2 time steps used during a MODFLOW time step when the tip or toe movement exceeds a user-defined value.

- **NADPTMX [-]:** Maximum number of SWI2 time steps per MODFLOW time step.
- **NADPTMN [-]:** Minimum number of SWI2 time steps per MODFLOW time step. NADPTMN must be less than or equal to NADPTMX.
- **ADPTFCT [-]:** ADPTFCT is the factor used to evaluate tip and toe thicknesses and control the number of SWI2 time steps per MODFLOW time step.

3.9.1.1.19. Time-Variant Specified Head (CHD)

For transient simulations, the Time-Variant Specified-Head (CHD) Package described in Appendix C of (Leake, et al., 1991) allows constant head cells to take on different head values for each time step. It provides a means of simulating head variations with time that can be approximated with piecewise linear segments. The CHD Package does not alter the way constant-head boundaries are formulated in MODFLOW.

The required input parameters for all input methods are identical and listed below.

1. **All input methods**⁵⁰: Figure 99 shows the dialog box for specifying cell, vertex, or polygon properties.

- **Flag [-]:** A non-zero value indicates that a cell is specified as a time-variant specified-head boundary.

- A no-flow cell cannot be converted to a CHD-cell.
- If a cell is specified as a time-variant specified-head boundary for a stress period and omitted in the specification for a subsequent period, it remains a constant-head boundary with a head equal to the end head of the last stress period for which it was specified.

- **Start Head (h_s) [L]⁵⁰ and End Head (h_e) [L]⁵⁰:** These values are the prescribed hydraulic head of the cell at the start and end of a stress period, respectively. For each time step within the stress period, the prescribed hydraulic head h at the cell is determined by

$$h = h_s + (h_e - h_s) \cdot \frac{PERTIM}{PERLEN} \quad (36)$$

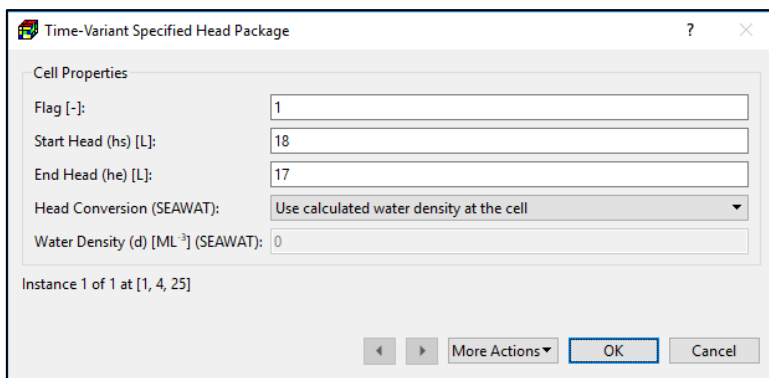


Figure 99. The Time-Variant Specified Head Package dialog box

⁵⁰ If multiple instances of CHD are assigned to a single cell, only one instance with the average values of h_s , h_e , and d of all instances will be used in the simulation.

where

$PERTIM$ is the starting time of the time step, and

$PERLEN$ is the length of the stress period.

The interpolated head values remain constant during the time step.

- **Head Conversion (SEAWAT):** This option determines how SEAWAT converts the specified values of h_s and h_e to reference head. Four options are available:
 - Use calculated water density at the cell
 - Use the specified water density value
 - Use the average of the calculated water density of the overlying cells
 - Input head values are reference head. No conversion
- **Water Density (d) [ML⁻³] (SEAWAT)⁵⁰:** Required only if the Head Conversion option is *Use the specified water density value*. SEAWAT uses this value to convert input head values to reference head.

3.9.1.1.20. Unsaturated-Zone Flow (UZF1)

The Unsaturated-Zone Flow (UZF1) Package (Niswonger, et al., 2006) was developed to simulate water flow and storage in the unsaturated zone and to partition flow into evapotranspiration and recharge. The package also accounts for land surface runoff to streams and lakes.

A kinematic wave approximation to Richards' equation is solved by the method of characteristics to simulate vertical unsaturated flow. The approach assumes that unsaturated flow occurs in response to gravity potential gradients only and ignores negative potential gradients; the approach further assumes uniform hydraulic properties in the unsaturated zone for each vertical column of model cells. The Brooks-Corey function is used to define the relation between unsaturated hydraulic conductivity and water content. Residual water content is calculated internally by the UZF1 Package based on the difference between saturated water content and specific yield.

The UZF1 Package is a substitution for the Recharge and Evapotranspiration Packages. The UZF1 Package differs from the Recharge Package in that an infiltration rate is applied at land surface instead of a specified recharge rate directly to ground water. The applied infiltration rate is further limited by the saturated vertical hydraulic conductivity. The UZF1 Package differs from the Evapotranspiration Package in that evapotranspiration losses are first removed from the unsaturated zone above the evapotranspiration extinction depth, and if the demand is not met, water can be removed directly from ground water whenever the depth to ground water is less than the extinction depth. The UZF1 Package also differs from the Evapotranspiration Package in that water is discharged directly to land surface whenever the altitude of the water table exceeds land surface.

The required input parameters for all input methods are identical and listed below. Although these parameters are to be assigned to each vertical column of model cells, you may move between layers within the Data Editor to examine the grid configuration in each layer.

1. **All input methods:** Figure 100 shows the dialog box for specifying cell, vertex, or polygon properties.

- **Data Tab**

- **UZF Option (NUZTOP) [-]:** Since recharge or discharge is calculated for only one cell in each vertical column of model cells, four options are available for specifying the cell where recharge or discharge is applied to:
 - 1. Recharge to and discharge from only the top model layer. This option assumes land surface is defined as top of layer 1.
 - 2. Recharge to and discharge from the layer IUZFBND and the land surface is defined as top of layer IUZFBND.
 - 3. Recharge to and discharge from the highest active cell in each vertical column and the land surface is defined as top of layer IUZFBND. A constant head cell intercepts any recharge and prevents deeper percolation.
 - 4. Recharge is added to the uppermost layer in a column that contains a water table. The layer to which recharge is added is updated at the beginning of each

The screenshot shows the 'Unsaturated-Zone Flow Package' dialog box with the 'Data' tab selected. The 'Simulation Settings' section has 'UZF Option (NUZTOP) [-]:' set to '3. Recharge to and discharge from highest active layers.'. The 'Package Data (Time-Independent)' section includes: 'Aerial Extent (IUZFBND) [-]:' 1, 'Overland Runoff (IRUNBND) [-]:' 0, 'Vertical Saturated Hydraulic Conductivity (VKS) [LT⁻¹):' 3, 'Brooks-Corey Epsilon (EPS) [-]:' 4, 'Saturated Water Content (THTS) [L³L⁻³):' 0.001, 'Residual Water Content (THTR) [L³L⁻³):' 0, and 'Initial Water Content (THTI) [L³L⁻³):' 0. The 'Cell Properties' section includes: 'Infiltration Rate (FINF) [LT⁻¹):' 1, 'Potential ET (PET) [LT⁻¹):' 0.13, 'ET Extinction Depth (EXTDP) [L]:' 20, 'ET Extinction Water Content (EXTWC) [L³L⁻³):' 0.0001, and 'Parameter Number (PEST) [-]:' 0. At the bottom, it shows 'Cell [Row, Column] = [132, 177]' and 'OK' and 'Cancel' buttons.

Figure 100. The Unsaturated-Zone Flow Package dialog box

new time step and is held constant during a time step. The values of IUZFBND are used to define the top of the unsaturated zone. This option is introduced in MODFLOW-NWT version 1.4 (released on March 1, 2020) and GSFLOW version 2.1 (released on April 6, 2020). This option should not be used with older MODFLOW versions.

○ **Package Data (Time-Independent)**

- **Aerial Extent (IUZFBND) [-]:** This value is used to define the aerial extent of the active model in which recharge and discharge will be simulated.
 - If IUZFBND = 0, recharge and discharge will not be simulated. IUZFBND should be set to 0 in no-flow (inactive) cells and in constant head cells that are defined by using a negative IBOUND value or through the FHB1 Package.
 - If IUZFBND < 0, the calculated infiltration will be added as recharge directly to groundwater, and storage changes in the unsaturated zone will be neglected.
 - If IUZFBND > 0, recharge and discharge are simulated according to the UZF Option (NUZTOP) defined above.
 - **Overland Runoff (IRUNBND) [-]:** This value is used to define the stream segments within the Streamflow-Routing (SFR2) Package or lake numbers in the Lake (LAK3) Package to which overland runoff from excess infiltration and groundwater discharge to land surface will be added. A positive integer value identifies the stream segment, and a negative integer value identifies the lake number.
 - **Vertical Saturated Hydraulic Conductivity (VKS) [LT⁻¹]:** This is the vertical saturated hydraulic conductivity of the UZF cell. If VKS is the same as VK, you can use *Tools > Data > Copy* to copy VK to VKS.
 - **Brooks-Corey Epsilon (EPS) [-]:** The epsilon exponent is used in the relation of water content to hydraulic conductivity.
 - **Saturated Water Content (THTS) [L³L⁻³]:** The saturated water content is approximately the same as the porosity.
 - **Residual Water Content (THTR) [L³L⁻³]:** the residual water content, commonly defined as the water content θ for which the gradient $d\theta/dh$ becomes zero, where h is the pressure head.
 - **Initial Water Content (THTI) [L³L⁻³]:** The water content at the beginning of the simulation. THTI is needed only if the first stress period is transient or if the SPECIFYTHTI box in the Options tab (see below) is checked.
- **Cell Properties** (for the Cell-by-Cell input method), **Polygon Properties** (for the Polygon input method), or **Vertex Properties** (for the Polyline input method)
- Infiltration Rate (FINF) [LT⁻¹]: This is the specified infiltration rate.
 - Potential ET (PET) [LT⁻¹]: Potential evapotranspiration flux.
 - ET Extinction Depth (EXTDP) [L]: The quantity of ET removed from a cell is limited by the volume of water stored in the unsaturated zone above the extinction depth. If groundwater is within the ET extinction depth, then the rate

removed is based on a linear decrease in the maximum rate at land surface and zero at the ET extinction depth.

- ET Extinction Water Content (EXTWC) [L^3L^{-3}]: The water content below which ET cannot be removed from the unsaturated zone.
- Parameter Number is used to group cells, where the FINF value is to be estimated by PEST.

- **Options Tab**

- **IUZFOPT**: Check this box to route water through unsaturated zone (i.e., IUZFOPT = 1 if checked; otherwise IUZFOPT = 0).
- **IRUNFLG**: Check this box to route ground water that discharges to land surface to stream segments or lakes as specified in the IRUNBND array.
- **IETFLG**: Specifies whether evapotranspiration (ET) will be simulated. ET will be simulated if this box is checked, otherwise it will not be simulated.
- **NTRAIL2 [-]**: This is the number of trailing waves used to define the water-content profile following a decrease in the infiltration rate.
- **NSETS2 [-]**: This is the number of wave sets used to simulate multiple infiltration periods.
- **SURFDEP [L]**: This is the average surface depression depth of the UZF cells.
- **NOSURFLEAK**: When the head is higher than the land surface, water seepage to the land surface may be removed from the groundwater system if this box is cleared.
- **SPECIFYTHTR**: If this box is checked, the residual water content (THTR) for each vertical column of cells is specified. Otherwise, the residual water content will be calculated.
- **SPECIFYTHTI**: If this box is checked, the initial water content (THTI) for each vertical column of cells is always specified, regardless of whether the first stress period is transient or steady-state.

3.9.1.1.21. Well (WEL)

The Well (WEL) Package is designed to simulate features such as wells that withdraw water from or add water to the aquifer at a constant rate during a stress period, where the rate is independent of both the cell area and the head in the cell. The Package can be used to simulate any features for which the recharge or discharge can be directly specified. Consider using the MNW1 Package (Section 3.9.1.1.10) for wells that penetrate multiple model layers. Refer to Section 3.6.21 for how to import wells from an external file.

The required input parameters are listed below. For large pumping data sets, consider using the **Tools > Package > Append Well** function described in Sections 3.6.21 and 5.1.18.

1. **All input methods** include a Simulation Settings group that contains the following option:
 - **Include Concentration Data in the SSM Package of MT3D/SEAWAT**: If this box is checked, the values entered to **Concentration (MT3D/SEAWAT) [ML^{-3}]** are used as

source water concentration. If this box is not checked, the concentration of the source water is assumed to be zero.

- **PHIRAMP:** PHIRAMP is the fraction of the cell thickness used as an interval for smoothly reducing pumping rates when the head in the cell is less than the interval above the cell bottom. Check the box to set PHIRAMP to the specified value and to save simulated pumping reduction data to the modflow.wellpumpingreduction file. If unchecked, the default value of 0.1 will be used. This option is only supported by in versions of MODFLOW that include the NWT solver, such as MODFLOW-NWT or GSFLOW.

2. **Cell-by-cell input method:** Figure 101 shows the dialog box for specifying cell properties.

- **Flow Rate (Q) [L³T⁻¹]:** Negative values are used to indicate pumping wells, while positive cell values indicate injection wells. The injection or pumping rate of a well is independent of both the cell area and the hydraulic head in the cell. MODFLOW assumes that a well penetrates the full thickness of the cell. To simulate wells that penetrate more than one model layer with the WEL Package, a flow rate must be specified for each penetrated layer. The total flow rate for a multi-layer well is equal to the sum of flow rates of individual layers.

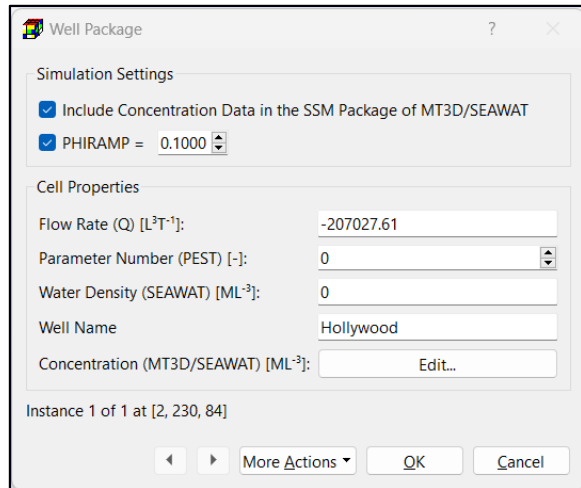


Figure 101. The Well Package dialog box for specifying cell properties

For confined layers, the flow rate for each layer Q_k can be approximately calculated by dividing the total rate (Q_{total}) in proportion to the layer transmissivities (McDonald, et al., 1988):

$$Q_k = Q_{total} \cdot \frac{T_k}{\Sigma T} \quad (37)$$

where T_k is the transmissivity of layer k, and ΣT is the sum of the transmissivity values of all layers penetrated by the multi-layer well.

Another possibility to simulate a multi-layer well with the WEL Package is to set a large vertical hydraulic conductivity (or vertical leakance), e.g. 1 m/s, to all cells of the well. The total pumping rate is then assigned to the lowest cell of the well. For display purposes, a very small pumping rate (e.g., $-1 \times 10^{-10} \frac{m^3}{s}$) can be assigned to other cells of the well. In this way, the exact extraction rate from each penetrated layer can be obtained by using ZoneBudget. See Section 0 for details about ZoneBudget.

- **Parameter Number** is used to group cells, where the Q value is to be estimated by PEST.

- **Water Density (SEAWAT) [ML⁻³]** is the density of injected water and is used by SEAWAT only if it is running in the uncoupled mode (i.e., the density-effect of all species are turned off, see Section 3.9.3.1) and WELDEN (see Figure 119. The VDF tab of the Transport Settings dialog box) is set as *User-Specified in the WEL Package*.
- **Well Name** is an optional text string to identify the well and will appear in the Input File to the Well Package. If you import wells from a Well File (see Section 5.1.18), the well names in that file will be imported and displayed here. To import wells from a Well Package input file that contains Well Names, add “AUX WELLNAME” to the end of the second line of the input file, the Well Names will be imported.
- **Concentration (MT3D/SEAWAT) [ML⁻³]**: Click the Edit button to open a Concentration dialog box (Figure 55) that allows you to specify the injected water concentration of up to 100 species. You can either enter individual concentration values to the table directly or click the *Set Value* button to enter a uniform value for all species. The entered concentration values may be used by the SSM Package of MT3D or SEAWAT.

3. **Polyline input method:** Figure 102 shows the dialog box for specify the following data to one or more vertices of a polyline. These vertex data can then be interpolated to model cells following the rules described in Section 3.4.2.2 and additional rules below.

- **Flow Rate per Unit Length [L²T⁻¹]** is interpolated to the cells along the polyline. For each of those cells, the interpolated cell value Q_l is multiplied with the length of the polyline within the cell to obtain the flow rate Q :

$$Q = Q_l \cdot L_c \tag{38}$$

where

Q_l is the flow rate per unit length at a given cell, and

L_c is the length of the polyline within the cell.

- **The other parameter values** are interpolated to cells along the polyline and the interpolated values are used as the input data to those cells.

4. **Polygon input method:** Figure 103 shows the dialog box for specify the following data are to a polygon. These data can then be assigned to model cells following the rules described in Section 3.4.2.3 and additional rules below.

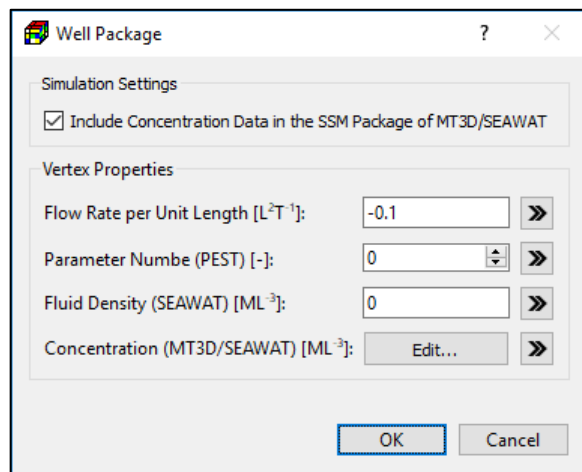


Figure 102. The Well Package dialog box for specifying vertex properties

- **Flow Rate per Unit Area [T^{-1}]** is assigned to the cells within the polygon. For each of those cells, the cell value Q_a is multiplied with the cell area to obtain the flow rate Q :

$$Q = Q_a \cdot Area_c \quad (39)$$

where

Q_a is the flow rate per unit area at a given cell, and

$Area_c$ is the area of the cell.

- **The other parameter values** are used as the input data to the cells within the polygon.

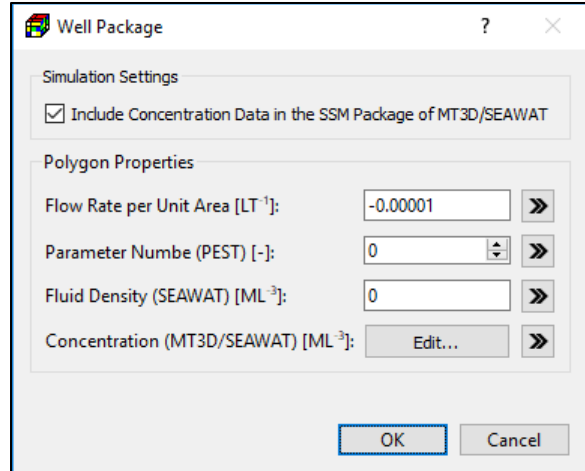


Figure 103. The Well Package dialog box for specifying polygon properties

3.9.1.1.22. Wetting Capability

If the BCF or LPF package is used and a model cell of an unconfined or convertible layer falls dry (i.e., its head falls below the cell bottom), the cell is set as a no-flow cell (with IBOUND = 0) and all conductance values to the cell are set to zero. As the simulation proceeds, a dry cell remains inactive even if the head at neighboring cells rise above its bottom.

When the Wetting Capability is activated, the BCF or LPF Package uses the Wetting Threshold (THRESH) to overcome this problem. At the start of a solution iteration, a dry cell may be converted to wet if $h \geq BOT + |THRESH|$, where h is the head value at an adjacent cell, BOT is the bottom elevation of the dry cell. When a dry cell is wetted, its IBOUND value is set to 1 (which indicates a variable-head cell), its original conductance value is restored, and its head value is set to a value slightly above the cell bottom. This mechanism allows dried cells to re-join the simulation.

The required input parameters for all input methods are identical and given below.

1. **All input method:** Figure 104 shows the dialog box for specifying cell, vertex, or polygon properties.
 - Interval (IWETIT) is the iteration interval for attempting to wet cells. Wetting is attempted every IWETIT iteration. If the PCG

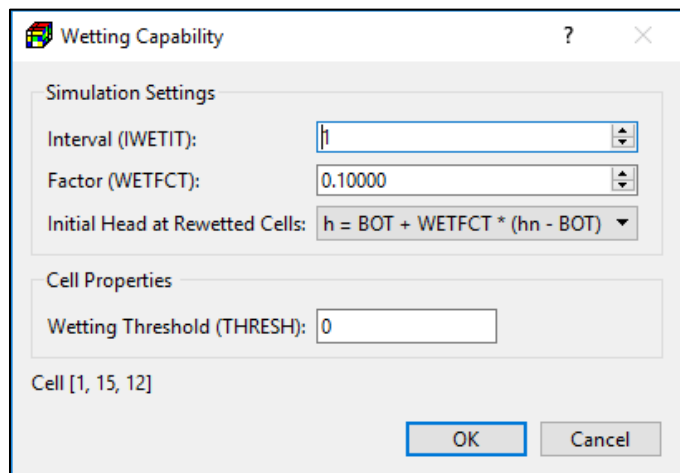


Figure 104. The Wetting Capability dialog box

solver is used, this applies to outer iterations, not inner iterations.

The reason for adjusting IWETIT is that the wetting of cells sometimes produces erroneous head changes in neighboring cells during the succeeding iteration, which may cause erroneous conversions of those cells. Waiting a few iterations until heads have had a chance to adjust before testing for additional conversions can prevent these errors. When setting IWETIT greater than one, there is some risk that cells may be prevented from correctly converting from dry to wet. If the solution for a time step is obtained in less than IWETIT iterations, then there will be no check during that time step to see if cells should be converted from dry to wet. The potential for this problem to occur is greater in transient simulations, which frequently require only a few iterations for a time step.

- **Factor (WETFCT)** is used in the calculation of the initial head at a cell that is converted from dry to wet.
- **Initial Head at Rewetted Cells:** This determines which equation is used to define the initial head (h) at the cell that becomes wet.

$$h = BOT + WETFCT \cdot (h_n - BOT) \quad (40)$$

$$h = BOT + WETFCT \cdot |THRESH| \quad (41)$$

where

BOT is the bottom elevation of the cell,

h_n is the head at an adjacent cell, and

$|THRESH|$ is the absolute value of the wetting threshold at the cell.

The initial estimate of head at a cell that converts to wet is theoretically unimportant because the solver should calculate the correct value in subsequent iterations. In practice, however, the initial estimate is often important for solver efficiency; that is, if a poor choice is made, the solver convergence may be slower than optimum. A bad choice for initial head also can cause the cell to oscillate between wet and dry.

- **Wetting Threshold (THRESH):** When the head at an adjacent cell equals or exceeds $BOT + |THRESH|$ of a dry cell, the dry cell is wetted. The sign of THRESH is used as a flag:
 - If $THRESH < 0$, only the cell below a dry cell can cause the cell to become wet.
 - If $THRESH > 0$, the cell below a dry cell and the four horizontally adjacent cells can cause a cell to become wet.
 - If $THRESH$ is 0, the cell cannot be wetted.

A primary difficulty with the rewetting approach is that the value of THRESH must be determined by trial and error. If $|THRESH|$ is too low, a cell may be incorrectly converted to wet. That is, a cell may convert to wet and then reconvert to dry in later solver iterations because the head does not stay above the bottom elevation. This cycle of converting between wet and dry conditions can repeat indefinitely, preventing the solver from converging. Larger values of $|THRESH|$ can avoid repeated conversion between wet and dry, but they increase the non-uniqueness of the solution (McDonald, et al., 1991).

The method of wetting and drying cells used in the BCF and LPF Packages can cause convergence problems with the iterative solvers. Symptoms of a convergence problem are slow convergence or divergence combined with the frequent wetting and drying of the same cells. For example, a well-cell with large pumping rate could cause the cell (and perhaps also neighboring cells) to fall dry. Thus, the pumping at that cell is switched off, the water table rises due to lack of pumping, the dry cell is wetted, and the pumping is re-established. The cycle repeats. Similar problems may occur near sinks or at places with small saturated thickness. You can detect such situations by examining the model run listing file with *Models > MODFLOW > View > Run Listing File*. A message is printed to that file each time a cell converts. Some options to mitigate the convergence problems are

1. Modify the settings
 - Set THRESH < 0, so that wetting can only be from below.
 - Increase IWETIT.
 - Use another equation for calculating the initial head at rewetted cells.
 - Increase the absolute value of THRESH.
2. Use a different solver or change solver settings.
3. Modify the vertical discretization such that only few cells will fall dry.
4. Decrease flow rates of sinks, such as pumping rates of wells.
5. Consider using MODFLOW-NWT or MODFLOW-OWHM with the UPW Package and Newton Solver.

3.9.1.2. Solvers

To calculate hydraulic heads, MODFLOW prepares one finite difference equation for each cell, expressing the relationship between the head at a node and the heads at each of the six adjacent nodes at the end of each time step. Because each equation may involve up to seven unknown values of head, and because the set of unknown head values changes from one equation to the next through the grid, the equations for the entire grid must be solved simultaneously at each time step. The system of simultaneous finite difference linear equations can be expressed in matrix notation as

$$\underline{A} \cdot \underline{x} = \underline{b} \quad (42)$$

where \underline{A} is a coefficient matrix assembled by MODFLOW using user-specified model data; \underline{b} is a vector of defined flows, terms associated with head-dependent boundary conditions and storage terms at individual cells; \underline{x} is a vector of hydraulic heads at individual cells. During a MODFLOW run, the vector \underline{x} is solved with one of the supported solvers.

3.9.1.2.1. Direct (DE4)

The Direct Solution (DE4) Package (Harbaugh, 1995) uses Gaussian elimination with an alternating diagonal equation numbering scheme that is more efficient than the standard

method of equation numbering. A direct solver is the most efficient when solving small, linear problems. Although direct solvers require more memory and typically require more computational effort than iterative solvers, the direct solver may execute faster than an iterative solver in some situations.

Figure 105 shows the dialog box for specifying the following required input parameters:

1. **Problem Type (IFREQ):** The choice of problem type affects the efficiency of solution; significant work can be avoided if it is known that the coefficient matrix \underline{A} (Equation 42) remains constant all or part of the time.

- **Linear** indicates that the flow equations are linear, but coefficients of simulated head for some stress terms may change at the start of each stress period. Examples of coefficients of simulated head for stress terms are riverbed conductance, drain conductance, maximum evapotranspiration rate, evapotranspiration extinction depth, and general-head boundary conductance.

- **Nonlinear** indicates that a non-linear flow equation is being solved, which means that some terms in \underline{A} depend on simulated head. Example of head dependent terms are transmissivity for water-table layers, which is based on the saturated thickness; flow terms for rivers, drains, and evapotranspiration convert between head dependent flow and constant flow; and the change in storage coefficient when a cell converts between confined and unconfined. When a non-linear flow equation is being solved, external iteration is normally required to accurately approximate the nonlinearities. Note that when nonlinearities caused by water table calculations are part of a simulation, there are not necessarily any obvious signs in the output from a simulation that does not use external iteration to indicate that iteration is needed. The budget error may be acceptably small without iteration even though there is significant error in head because of non-linearity. To understand this, consider the water table correction for transmissivity. For each iteration, a new transmissivity value is

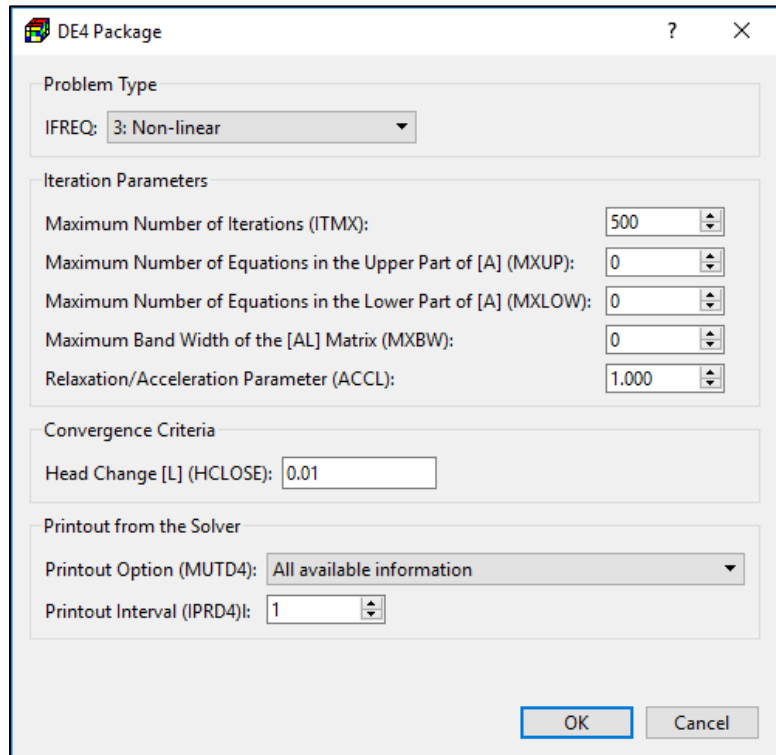


Figure 105. The DE4 Package dialog box

calculated based on the previous head. Then the flow equations are solved, and a budget is computed using the new head with the same transmissivities. No budget discrepancy results because heads are correct for the transmissivity being used at this point; however, the new heads may cause a significant change in transmissivity. The new transmissivity will not be calculated unless there is another iteration. Therefore, when one or more layers are under water table conditions, iteration should always be tried. The maximum change in head during each iteration (printed by the solver) provides an indication of the impact of all nonlinearities.

2. Iteration Parameters

- **Maximum Number of Iterations (ITMX)** is the maximum number of iterations in each time step. Specify $ITMX = 1$ if iteration is not desired. Ideally iteration would not be required for direct solution; however, it is necessary to iterate if the flow equation is nonlinear or if computer precision limitations result in inaccurate calculations as indicated by a large water budget error.
- **Maximum Number of Equations in the Upper Part of [A] (MXUP)** is the maximum number of equations in the upper part of the equations to be solved. This value impacts the amount of memory used by the solver. If specified as 0, the solver will calculate MXUP as half the number of cells in the model, which is an upper limit.
- **Maximum Number of Equations in the Lower Part of [A] (MXLOW)** is the maximum number of equations in the lower part of the equations to be solved. This value impacts the amount of memory used by the solver. If specified as 0, the solver will calculate MXLOW as half the number of cells in the model, which is an upper limit.
- **Maximum Band Width of the [AL] Matrix (MXBW)** is the maximum band width plus 1 of the [AL] matrix. This value impacts the amount of memory used by the solver. If specified as 0, the solver will calculate MXBW as the product of the two smallest grid dimensions plus 1, which is an upper limit.
- **Relaxation/Acceleration Parameter (ACCL)** is a multiplier for the computed head change for each iteration. Normally this value is 1. A value greater than 1 may be useful for improving the rate of convergence when using external iteration to solve non-linear problems. ACCL should always be 1 for linear problems. When $ITMX=1$, the solver changes ACCL to 1 regardless of the input value.

3. Convergence Criteria

- **Head Change [L] (HCLOSE):** This is the head change criterion for convergence. When the maximum absolute value of head change from all nodes during an iteration is less than or equal to HCLOSE, iteration stops.

4. Printout from the Solver

- **Printout Option (MUTD4):** This controls printing of convergence information from the solver.
 - **All available information:** The maximum head change and residual (positive or negative) are saved in the run listing file for each iteration of a time step whenever the time step is an even multiple of Printout Interval.

- **The number of iterations only:** The printout of maximum head change and residual is suppressed.
- **None:** Suppress all printout from the solver.
- **Printout Interval (IPRD4):** This is the time step interval for printing out convergence information when iterating ($ITMX > 1$).

3.9.1.2.2. Strongly Implicit Procedure (SIP)

Figure 106 shows the dialog box for specifying the following required input parameters:

1. Iteration Parameters

- **Maximum Number of Iterations (MXITER)** is the maximum number of times through the iteration loop within a time step to solve the system of equations.
- **Number of Iteration Parameters (NPARM)** is the number of iteration variables to be used. Five variables are generally sufficient.
- **Acceleration Parameter (ACCL)** is the acceleration variable. It must be greater than zero and is generally equal to one. If a zero is entered, it is changed to one by the solver.

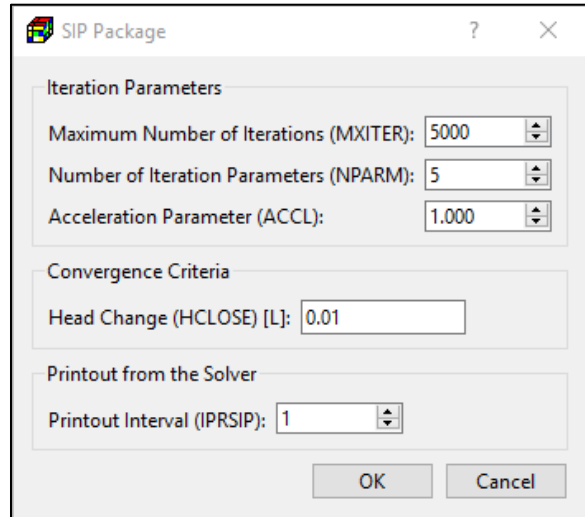


Figure 106. The SIP Package dialog box

2. Convergence Criteria

- **Head Change (HCLOSE) [L]:** This is the head change criterion for convergence. When the maximum absolute value of head change from all nodes during an iteration is less than or equal to HCLOSE, iteration stops.

3. Printout from the Solver

- **Printout Interval (IPRSIP):** The maximum head change (positive or negative) is printed for each iteration of a time step whenever the time step is an even multiple of IPRSIP.

3.9.1.2.3. Geometric Multigrid (GMG)

Figure 107 shows the dialog box for specifying the following required input parameters of the Geometric Multigrid Solver (GMG) Package (Wilson, et al., 2004).

1. Preconditioning Method

- **Smoother Type (ISM):** ISM controls the type of smoother used in the multigrid preconditioner.

- **ILU Smoothing:**

Select this option to implement ILU(0) smoothing in the multigrid preconditioner.

This smoothing requires an additional vector on each multigrid level to store the pivots in the ILU factorization.

- **Symmetric Gauss-Seidel (SGS) Smoothing:**

Select this option to implement the Symmetric Gauss-Seidel (SGS) smoothing in the multigrid preconditioner. No

additional storage is required for this smoother; users may want to use this option if available memory is exceeded or nearly exceeded when using ILU Smoothing. Using SGS smoothing is not as robust as ILU smoothing; additional iterations are likely to be required in reducing the residuals. In extreme cases, the solver may fail to converge as the residuals cannot be reduced sufficiently.

- **Semicoarsening (ISC):** This option controls semi-coarsening in the multigrid preconditioner. The available options and their meanings are given as follows:
 - The rows, columns and layers are all coarsened.
 - The rows and columns are coarsened, but the layers are not.
 - The columns and layers are coarsened, but the rows are not.
 - The rows and layers are coarsened, but the columns are not.
 - No coarsening takes place.

Typically, the first or second option should be selected. In the case that there are large vertical variations in the hydraulic conductivities, then the second option should be used. If no coarsening is implemented, the GMG solver is comparable to the PCG ILU(0) solver and uses the least amount of memory.

- **Relaxation Parameter (RELAX)** is a relaxation parameter for the ILU preconditioned conjugate gradient method. The RELAX parameter can be used to improve the spectral condition number of the ILU preconditioned system. The value of RELAX

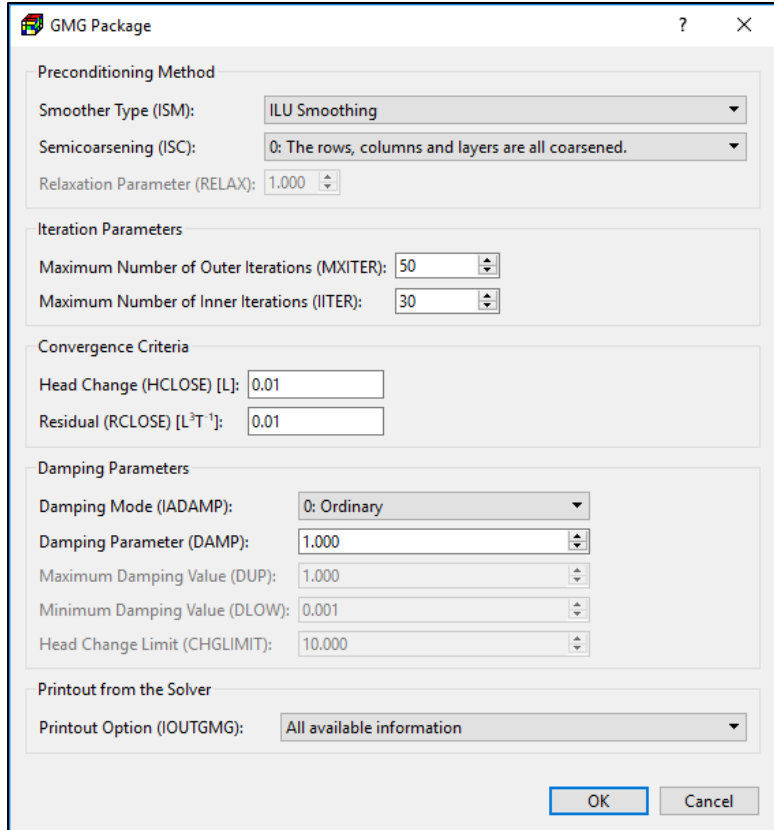


Figure 107. The GMG Package dialog box

should be approximately one. However, the relaxation parameter can cause the factorization to break down. If this happens, then the GMG solver will report an assembly error and a value smaller than one for RELAX should be tried.

2. Iteration Parameters

- **Maximum Number of Outer Iterations (MXITER):** For linear problems, MXITER can be set to 1. For nonlinear problems, MXITER needs to be larger, but rarely more than 100.
- **Maximum Number of Inner Iterations (IITER)** is the maximum number of PCG iterations for each linear solution. A value of 100 is typically sufficient. It is frequently useful to specify a smaller number for nonlinear problems to prevent an excessive number of inner iterations.

3. Convergence Criteria

- **Head Change (HCLOSE) [L]** is the head change convergence criterion for nonlinear problems. After each linear solve (inner iteration), the maximum of the head change is compared against HCLOSE. HCLOSE can be set to a large number for linear problems; HCLOSE is ignored if MXITER=1.
- **Residual (RCLOSE) [L³T⁻¹]** is the residual convergence criterion for the inner iteration. Typically, RCLOSE is set to the same value as HCLOSE. If RCLOSE is set too high, then additional outer iterations may be required due to the linear equation not being solved with sufficient accuracy. On the other hand, a too restrictive setting for RCLOSE for nonlinear problems may force an unnecessarily accurate linear solution.

4. Damping Parameters

- **Damping Mode (IADAMP)** is a flag that controls adaptive damping. The possible values of IADAMP and their meanings are as follows:
 - **0: Ordinary:** the value assigned to DAMP is used as a constant damping parameter.
 - **1: Cooley's method:** the value of DAMP is used for the first nonlinear iteration. The damping parameter is adaptively varied based on the head change, using Cooley's method as described in (Mehl, et al., 2001), for subsequent iterations.
 - **2. Relative reduced residual method:** the relative reduced residual damping method documented in (Mehl, et al., 2001) and modified by (Banta, 2006) is used.

When IADAMP is specified as 2 and the value specified for DAMP is less than 0.5, the closure criterion for the inner iterations (DRCLOSE) is assigned simply as RCLOSE. When DAMP is between 0.5 and 1.0, inclusive, or when IADAMP is specified as 0 or 1, DRCLOSE is calculated according to equation 20 on p. 9 of (Wilson, et al., 2004)

- **Damping Parameter (DAMP)** is the value of the damping parameter. For linear problems, a value of 1.0 should be used. For nonlinear problems, a value less than 1.0 but greater than 0.0 may be necessary to achieve convergence. A typical value for nonlinear problems is 0.5. Damping also helps control the convergence criterion of the linear solve to alleviate excessive PCG iterations.
- **Maximum Damping Value (DUP)** is the maximum damping value that should be applied at any iteration when the solver is not oscillating; it is dimensionless. An

appropriate value for DUP will be problem dependent. For moderately nonlinear problems, reasonable values for DUP would be in the range 0.5 to 1.0. For a highly nonlinear problem, a reasonable value for DUP could be as small as 0.1. When the solver is oscillating, a damping value as large as $2.0 \times \text{DUP}$ may be applied.

- **Minimum Damping Value (DLOW)** is the minimum damping value to be generated by the adaptive-damping procedure; it is dimensionless. An appropriate value for DLOW will be problem-dependent and will be smaller than the value specified for DUP. For a highly nonlinear problem, an appropriate value for DLOW might be as small as 0.001.
- **Head Change Limit (CHGLIMIT)** is the maximum allowed head change at any cell between outer iterations.

5. Printout from the Solver

- **Printout Option (IOUTGMG):** This controls printing of convergence information from the solver. Available options are
 - Only the solver inputs
 - All available information
 - The convergence history of the PCG iteration

3.9.1.2.4. Preconditioned Conjugate Gradient (PCG)

The Preconditioned Conjugate-Gradient Solver (PCG) Package is fully documented in (Hill, 1990). The PCG solver incorporates two forms of the Preconditioned Conjugate-Gradient method: the Modified Incomplete Cholesky and the Polynomial methods. Knowledge of the details of these methods generally is not necessary. The most important aspect is that the Modified Incomplete Cholesky method is almost always faster. The Polynomial method is potentially faster only on parallel computers where some of the work can be performed simultaneously on multiple processors.

Figure 108 shows the dialog box for specifying the following required input parameters:

1. Preconditioning Method

- **NPCOND:** This is the matrix preconditioning method. The available options are
 - Modified Incomplete Cholesky (for use on scalar computers)
 - Neuman series polynomial (for use on parallel computers)
- **NBPOL:** This is used with the Neuman series polynomial preconditioning method. Checking this box to calculate the estimate of the upper bound on the maximum eigenvalue. If this box is cleared, the estimate is set as 2. Convergence is generally insensitive to this parameter.

2. Iteration Parameters

- **Maximum Number of Outer Iterations (MXITER):** For a linear problem MXITER should be 1, unless more than 50 inner iterations are required, when MXITER could be as large as 10. A larger number (generally less than 100) is required for a nonlinear problem.

For a linear problem, a single outer iteration with many inner iterations theoretically is adequate, but a few outer iterations should be used for large (greater than 10,000 cells) linear models. For numerical reasons described in p. 8 of (Hill, 1990), the outer iterations improve the accuracy of large linear problems.

- **Maximum Number of Inner Iterations (ITERI):**

For nonlinear problems, ITERI usually ranges from 10 to 30; a value of 30 will be sufficient for most linear problems.

Nonlinear problems generally run most efficiently with five or more outer iterations and a large number of inner iterations (20 or more). The solution process becomes inefficient if the nonlinear terms are reformulated frequently by having a small (less than 20) value for ITERI.

- **Relaxation Parameter (RELAX):** This is the relaxation parameter used with the Modified Incomplete Cholesky preconditioning method. Usually, RELAX=1.0, but for some problems a value of 0.99, 0.98, or 0.97 will reduce the number of iterations required for convergence.
- **Damping Parameter (DAMPPCG):** Values less than 1.0 can be used to aid convergence in nonlinear problems.

3. Convergence Criteria

- **Head Change (HCLOSE) [L]** is the head change criterion for convergence. When the maximum absolute value of head change from all nodes during an iteration is less than or equal to HCLOSE, and the criterion for RCLOSE is also satisfied (see below), iteration stops.
- **Residual (RCLOSE) [L^3T^{-1}]** is the residual criterion for convergence. When the maximum absolute value of the residual at all nodes during an iteration is less than or equal to RCLOSE, and the criterion for HCLOSE is also satisfied (see above), iteration stops. For nonlinear problems, convergence is achieved when the convergence criteria are satisfied for the first inner iteration.

4. Printout from the Solver

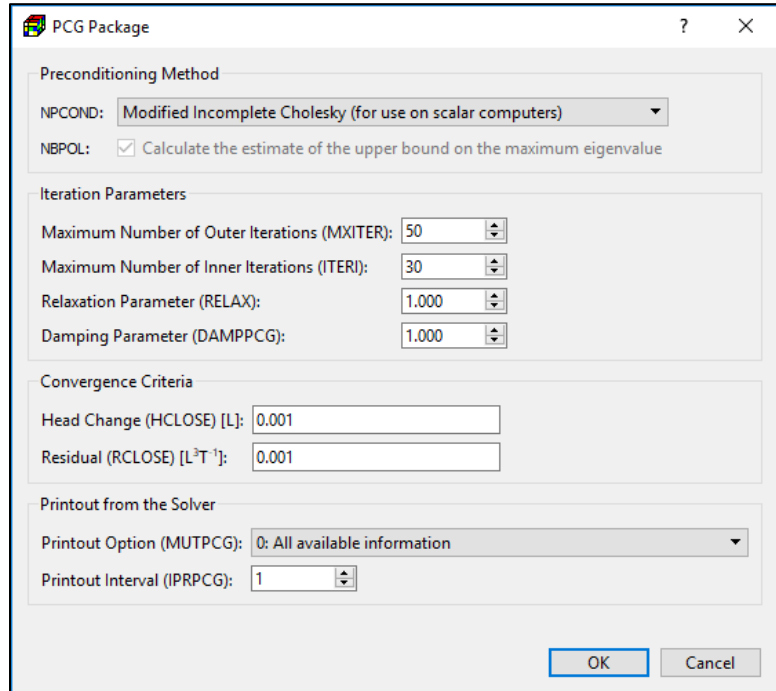


Figure 108. The PCG Package dialog box

- **Printout Option (MUTPCG)** is a flag that controls printing of convergence information from the solver:
 - **0: All available information:** Print tables of maximum head change and residual each iteration.
 - **1: The number of iterations only:** Print only the total number of iterations
 - **2: None:** No Printing.
 - **3:** Print as for 0, but only if solver convergence fails
- **Printout Interval (IPRPCG):** The extreme head change and residual (positive or negative) are printed for each iteration of a time step whenever the time step is an even multiple of the printout interval. The printout also occurs at the end of each stress period regardless of the value of the printout interval.

3.9.1.2.5. PCG with Improved Nonlinear Control (PCGN)

The principal objective of the preconditioned conjugate gradient with improved nonlinear control (PCGN) Package (Naff, et al., 2008) is to provide the modeler with more options when faced with a poorly converging nonlinear problem. Picard iteration is used to solve nonlinear ground-water flow equations by iteratively solving a linear approximation of the nonlinear equations. The linear solution is provided by means of the preconditioned conjugate gradient algorithm where preconditioning is provided by the modified incomplete Cholesky algorithm. The incomplete Cholesky scheme incorporates two levels of fill, 0 and 1, in which the pivots can be modified so that the row sums of the preconditioning matrix and the original matrix are approximately equal. A relaxation factor is used to implement the modified pivots, which determines the degree of modification allowed. The preconditioned conjugate gradient method is coupled with Picard iteration to efficiently solve the nonlinear equations associated with many ground-water flow problems.

Figure 109 shows the dialog box for specifying the following required input parameters:

1. Iteration Parameters

- **Maximum Number of Outer Iterations (ITER_MO):** For nonlinear problems, this variable must be set to some number greater than one, depending on the problem size and degree of nonlinearity. If this variable is set to 1, then the PCGN solver assumes that the problem is linear.
- **Maximum Number of Inner Iterations (ITER_MI):** Generally, this variable is set to some number greater than one, depending on the matrix size, degree of convergence called for, and the nature of the problem. For a nonlinear problem, this variable should be set large enough

- **Relaxation Parameter (RELAX)** is used with the modified incomplete Cholesky preconditioning method. A value of RELAX between 0.9 and 1 is generally advised for most problems.
- **Fill Level of the MIC Preconditioner (IFILL):** Generally, the higher the fill level, the more preconditioning imparted by a modified incomplete Cholesky preconditioner. For problems where the matrix equation is not well behaved or for a nonlinear problem where convergence is not easily achieved, a fill level of 1 may provide the additional preconditioning necessary to obtain convergence.

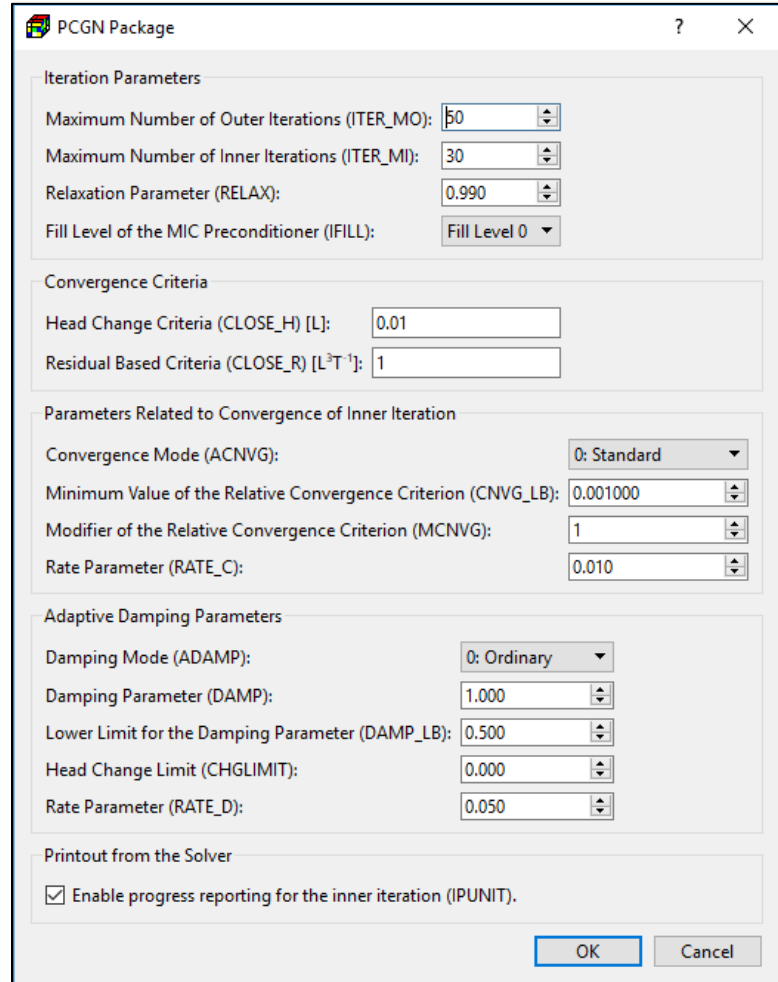


Figure 109. The PCGN Package dialog box

2. Convergence Criteria

- **Residual Based Criteria (CLOSE_R) [L³T⁻¹]:** This parameter is used differently, depending on whether it is applied to a linear or nonlinear problem:
 - **ITER_MO = 1:** CLOSE_R is used as the value in the absolute convergence criterion for quitting the PCG iterative solver. CLOSE_R is compared to the square root of the weighted residual norm v . This norm is defined as $v = r^T M^{-1} r$, where M represents the preconditioning matrix with the PCG algorithm and r is a vector of residuals. If \sqrt{v} is less than CLOSE_R, then the linear PCG iterative solve is said to have converged.
 - **ITER_MO > 1:** For a nonlinear problem, CLOSE_R is used as a criterion for quitting the Picard (outer) iteration. CLOSE_R is compared to the square root of the inner product of the residuals (the residual norm), $[r^T r]^{1/2}$, as calculated on entry to the PCG solver at the beginning of every Picard iteration. If this norm is less than CLOSE_R, then the Picard iteration is considered to have converged.

- **Head Change Criteria (CLOSE_H) [L]:** CLOSE_H is used as an alternate stopping criterion for the Picard iteration needed to solve a nonlinear problem. The maximum value of the head change is obtained for each Picard iteration, after completion of the inner, PCG iteration. If this maximum head change is less than CLOSE_H, then the Picard iteration is considered tentatively to have converged. However, as nonlinear problems can demonstrate oscillation in the head solution, the Picard iteration is not declared to have converged unless the maximum head change is less than CLOSE_H for three Picard iterations. If these Picard iterations are sequential, then a good solution is assumed to have been obtained. If the Picard iterations are not sequential, then a warning is issued advising that the convergence is conditional, and the user is urged to examine the mass balance of the solution.

3. Parameters Related to Convergence of Inner Iteration

- **Convergence Mode (ACNVG):** ACNVG defines the mode of convergence applied to the PCG solver. In general, the relative stopping criterion for PCG iteration is $v_i < \varepsilon v_0$ where v_0 is the weighted residual norm on entry to the PCG solver, ε is the relative convergence parameter, and v_i is the same norm at PCG iteration i . The available convergence modes are:
 - **ACNVG=0:** The standard ε convergence scheme is employed. The standard relative convergence is denoted by ε_s and usually takes the value 0.1; this value is assigned to the relative convergence ε . No additional variables are used.
 - **ACNVG=1:** Adaptive convergence is employed. The adaptive convergence scheme adjusts the relative convergence ε of the PCG iteration based on a measure of the nonlinearity of the problem. Under this scheme, ε is allowed to vary such that $\text{CNVG_LB} < \varepsilon < \varepsilon_s$ where the exact value of ε is dependent on the measure of nonlinearity. This option requires a valid value for variable CNVG_LB.
 - **ACNVG=2:** Enhanced convergence is employed. If the variable enhancement option is employed ($\text{RATE_C} > 0$), then ε_s is taken as the upper limit for ε . This option requires valid values for variables MCNVG and RATE_C.
- **Minimum Value of the Relative Convergence Criterion (CNVG_LB)** is used only in convergence mode ACNVG = 1. CNVG_LB is the minimum value that the relative convergence ε can take under the self-adjusting convergence option. The objective here is to prevent ε from becoming so small that the PCG solver takes an excessive number of iterations. Valid range for variable: $0 < \text{CNVG_LB} < \varepsilon_s$; a value of CNVG_LB = 0.001 usually produces reasonable results.
- **Modifier of the Relative Convergence Criterion (MCNVG)** is used only in convergence mode ACNVG = 2. MCNVG increases the relative PCG convergence criteria by a power equal to MCNVG; that is, letting $p = \text{MCNVG}$, then the relative convergence criterion ε is enhanced such that $\varepsilon = \varepsilon_s^p$ where $0 < p \leq 6$. If MCNVG is set to a value greater than 6, then PCGN resets MCNVG = 6 and issues a warning message. If RATE_C = 0, then this enhanced relative convergence criterion is applied uniformly throughout the simulation; the relative convergence, in this case, is not adjusted for stress changes in

the simulation. MCNVG must be set to a value greater than zero when ACNVG = 2; otherwise a data error is declared and ACNVG is reset to zero.

- **Rate Parameter (RATE_C)** is used only in convergence mode ACNVG = 2; this option results in variable enhancement of ε . If $0 < \text{RATE_C} < 1$, then enhanced relative convergence is allowed to decrease by increasing ε as follows: $\varepsilon_j = \varepsilon_{j-1} + \text{RATE_C} \cdot \varepsilon_{j-1}$ where j is the Picard iteration number; this change in ε occurs so long as the Picard iteration progresses satisfactorily. If $\text{RATE_C} \leq 0$, then the value of ε set by MCNVG remains unchanged through the Picard iteration. It should be emphasized that RATE_C must have a value greater than 0 for the variable enhancement to be effective; otherwise ε remains constant. The assumption here is that a better solution of the linear equations is needed initially to improve the nonlinear Picard solution, but that this need abates with additional Picard iterations. Typical values for RATE_C are $0.01 < \text{RATE_C} < 0.1$, although larger or smaller values may be used.

4. Adaptive Damping Parameters

- **Damping Mode (ADAMP)** defines the mode of damping applied to the linear solution. In general, damping determines how much of the head changes vector Δ_j shall be applied to the hydraulic head vector h_j in Picard iteration j : $h_j = h_{j-1} + \theta \Delta_j$, where θ is the damping parameter. The available damping modes are:
 - **ADAMP = 0:** Ordinary damping is employed and a constant value of damping parameter $\theta = \text{DAMP}$ will be used throughout the Picard iteration. This option requires a valid value for DAMP.
 - **ADAMP = 1:** Adaptive damping is employed; Adaptive damping changes the damping parameter θ in response to the difficulty the nonlinear solver encounters in solving a given problem. Essentially, the nonlinear solver looks to increase θ should the convergence of the Picard iteration proceed satisfactorily, but otherwise causes θ to decrease. Adaptive damping can be useful for problems that do not converge readily, but otherwise should be avoided as it generally requires more total iterations. This option requires valid values for variables DAMP, DAMP_LB, RATE_D, and CHGLIMIT. Adaptive damping also admits the possibility of directly limiting the maximum head change applicable to update the hydraulic heads; see CHGLIMIT below. If this option is not desired, then CHGLIMIT should be set to zero.
 - **ADAMP = 2:** Enhanced damping algorithm in which the value of θ is increased (but never decreased) provided the Picard iteration is proceeding satisfactorily. This enhanced damping allows θ to increase from a minimum value to a maximum value DAMP by a rate equal to RATE_D. The minimum value in the first stress period is DAMP_LB; for subsequent stress periods, it is the geometric mean of DAMP and DAMP_LB. This option requires valid values for DAMP, DAMP_LB, and RATE_D.
- **Damping Parameter (DAMP)** restricts the damping parameter θ ; generally, $0 < \text{DAMP} < 1$. Its function for the various modes of ADAMP are:

- **ADAMP = 0:** The damping parameter θ takes on the value DAMP and is maintained constant throughout the simulation.
- **ADAMP > 0:** The value of DAMP will be treated as the upper limit for θ in the enhanced damping or adaptive damping algorithms.
- **Lower Limit for the Damping Parameter (DAMP_LB)** represents a bound placed on θ ; generally, $0 < \text{DAMP_LB} < \text{DAMP}$. For the various modes of $\text{ADAMP} > 0$, DAMP_LB serves the following purposes:
 - **ADAMP = 1:** In the adaptive damping algorithm, DAMP_LB represents the lower limit to which θ , under adverse adaptive damping conditions, will be allowed to fall.
 - **ADAMP = 2:** In the enhanced damping algorithm, DAMP_LB is the starting value (or a component of the starting value) for the damping parameter θ used in the initial Picard iteration of every stress period.
- **Head Change Limit (CHGLIMIT)** limits the maximum head change applicable to the updated hydraulic heads in a Picard iteration. Provided that the current damping factor is greater than the ratio of CHGLIMIT to the maximum head change and that this ratio is less than one, then the damping factor is reset to the value of the ratio. This option is available only in association with adaptive damping: $\text{ACNVG} = 1$. If $\text{CHGLIMIT} = 0.0$, then adaptive damping proceeds without this feature.
- **Rate Parameter (RATE_D):** Generally, $0 < \text{RATE_D} < 1$. For the various modes of $\text{ADAMP} > 0$, RATE_D serves the following purposes:
 - **ADAMP = 1:** RATE_D sets the recovery rate for the damping factor θ in response to the progress in the Picard iteration; it also forms a limit on the response function to progress in the Picard iteration. Typical values for RATE_D, under this scenario, are $0.01 < \text{RATE_D} < 0.1$. Under adaptive damping, if the user finds that the damping factor θ increases too rapidly, then reducing RATE_D will slow the rate of increase.
 - **ADAMP = 2:** Provided the Picard iteration is progressing satisfactorily, RATE_D adjusts the damping factor θ upward such that $\theta_j = \theta_{j-1} + \text{RATE_D} \cdot \theta_{j-1}$, where j is the Picard iteration number. Typical values for RATE_D, under this scenario, are $0.01 < \text{RATE_D} < 0.1$, although larger or smaller values may be used.

5. Printout from the Solver

- Enable progress reporting for the inner iteration (IPUNIT): If checked, a record of progress made by the Picard iteration for each time step is printed in the Run Listing file. This record includes the total number of dry cells at the end of each time step as well as the total number of PCG iterations necessary to obtain convergence.

3.9.1.2.6. Newton Solver (NWT)

This package is activated only if the Upstream Weighting (UPW) Package is selected as the groundwater flow package. See Section 1.3 for details about groundwater flow package.

Figure 110 shows the dialog box for specifying the following required input parameters:

1. Solution Options

- **LINMETH:** This determines which matrix solver will be used.
- **OPTIONS:** This determines which default solver input values will be used. See table 2 of the MODFLOW-NWT user guide (Niswonger, et al., 2011) for the solver input values that will be used for the available options.
 - **SIMPLE (nearly linear model)** indicates that default solver input values will be defined that work well for nearly linear models. This would be used for models that do not include nonlinear stress packages, and models that are either confined or consist of a single unconfined layer that is thick enough to contain the water table within a single layer.
 - **MODERATE (moderately nonlinear model)** indicates that default solver input values will be defined that work well for moderately nonlinear models. This would be used for models that include nonlinear stress packages, and models that consist of one or more unconfined layers. The MODERATE option should be used when the SIMPLE option does not result in successful convergence.
 - **COMPLEX (highly nonlinear model)** indicates that default solver input values will be defined that work well for highly nonlinear models. This would be used for models that include nonlinear stress packages, and models that consist of one or more unconfined layers representing complex geology and surface water and groundwater interaction. The COMPLEX option should be used when the MODERATE option does not result in successful convergence.
 - **SPECIFIED (use optional parameters specified below)** indicates that the optional solver input parameters will be specified in the Optional Parameters section of this dialog box.
- **IBOTAV** is a flag that indicates whether corrections will be made to groundwater head relative to the cell-bottom altitude if the cell is surrounded by dewatered cells. A correction will be made if this box is checked. This setting is problem specific and both checked and unchecked should be tested.

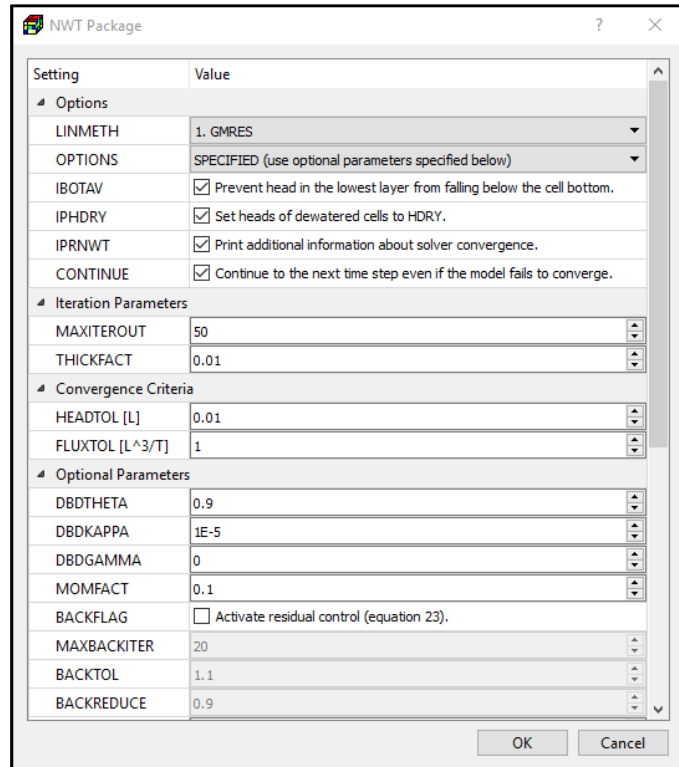


Figure 110. The NWT Package dialog box

- **IPHDRY** is a flag that indicates whether groundwater head will be set to HDRY when the groundwater head is less than 1E-04 above the cell bottom.
- **IPRNWT** is a flag that indicates whether additional information about solver convergence will be printed to the run listing file.
- **CONTINUE**: Check this box to allow the model simulation to continue to the next step even if the model failed to converge.

2. Iteration Parameters

- **MAXITEROUT** is the maximum number of iterations to be allowed for solution of the outer (nonlinear) problem.
- **THICKFACT** is the portion of the cell thickness (length) used for smoothly adjusting storage and conductance coefficients to zero.

3. Convergence Criteria

- **Head Change (HEADTOL) [L]** is the maximum head change between outer iterations for solution of the nonlinear problem.
- **Residual (FLUXTOL) [L³T⁻¹]** is the maximum root-mean-squared flux difference between outer iterations for solution of the nonlinear problem.

4. Optional Parameters. The optional parameters are used only if OPTIONS is set to SPECIFIED. See Table 2 of (Niswonger, et al., 2011) for a list of suggested values for these parameters.

- **DBDTHETA** is a coefficient, symbol θ in equation 21 of (Niswonger, et al., 2011), used to reduce the weight applied to the head change between nonlinear iterations. DBDTHETA is used to control oscillations in head. Values range between 0.0 and 1.0, and larger values increase the weight (decrease under-relaxation) applied to the head change.
- **DBDKAPPA** is a coefficient, symbol κ in equation 22 of (Niswonger, et al., 2011), used to increase the weight applied to the head change between nonlinear iterations. DBDKAPPA is used to control oscillations in head. Values range between 0.0 and 1.0, and larger values increase the weight applied to the head change.
- **DBDGAMMA** is a factor, symbol γ in equation 19 of (Niswonger, et al., 2011), used to weight the head change for iterations $n-1$ and n . Values range between 0.0 and 1.0, and greater values apply more weight to the head change calculated during iteration n .
- **MOMFACT** is the momentum coefficient m of equation 20 and ranges between 0.0 and 1.0. Greater values apply more weight to the head change for iteration n .
- **BACKFLAG** is a flag used to specify whether residual control will be used. A value of 1 indicates that residual control is active and a value of 0 indicates residual control is inactive.
- **MAXBACKITER** is the maximum number of reductions (backtracks) in the head change between nonlinear iterations. A value between 10 and 50 works well. **MAXBACKITER** is enabled and used only if **BACKFLAG** is checked.

- **BACKTOL** is the proportional decrease in the root-mean-squared error of the groundwater-flow equation used to determine if residual control is required at the end of a nonlinear iteration, as applied in equation 23 of (Niswonger, et al., 2011). **BACKTOL** is enabled and used only if **BACKFLAG** is checked.
- **BACKREDUCE** is a reduction factor, symbol B_r in equation 23 of (Niswonger, et al., 2011), used for residual control that reduces the head change between nonlinear iterations. Values should be between 0.0 and 1.0, where smaller values result in smaller head-change values. **BACKREDUCE** is enabled and used only if **BACKFLAG** is checked.
- The following parameters are enabled and used only if **LINMETH** is **1. GMRES**.
 - **MAXITINNER** is the maximum number of inner iterations for the linear solution.
 - **ILUMETHOD** is the method for incomplete factorization (ILU) used as a preconditioner.
 - **LEVFILL** is the fill limit for **ILUMETHOD** = 1 and is the level of fill for **ILUMETHOD** = 2. Recommended values: 5-10 for method 1, 0-2 for method 2.
 - **STOPTOL** is the tolerance for convergence of the linear solver. This is the residual of the linear equations scaled by the norm of the root mean squared error. Usually 10^{-8} to 10^{-12} works well.
 - **MSDR** is the number of iterations between restarts of the GMRES Solver.
- The following parameters are enabled and used only if **LINMETH** is **"2. χMD"**.
 - **IACL** is the acceleration method.
 - **NORDER** is the scheme of ordering the unknowns.
 - **LEVEL** is the level of fill for incomplete LU factorization.
 - **NORTH** is the number of orthogonalization for the ORTHOMIN acceleration scheme. A number between 4 and 10 is appropriate. Small values require less storage, but more iterations may be required. This number should equal 2 for the other acceleration methods.
 - **IREDSYS** decides whether to apply reduced system preconditioning.
 - **RRCTOLS** is the residual reduction-convergence criteria.
 - **IDROPTOL** decides whether to use drop tolerance in the preconditioning.
 - **EPSRN** is the drop tolerance for preconditioning.
 - **HCLOSEXMD [L]** is the head closure criteria for inner (linear) iterations.
 - **MXITERXMD** is the maximum number of inner iterations for the linear solution.

3.9.1.3. Output Control

The primary output file of MODFLOW is the run listing file that can be found in *Models > Modflow > View > Run Listing File*. In addition to the run listing file, various simulation results can be saved by checking the corresponding output terms in the Modflow Output Control dialog box (Figure 111). The available settings are described below.

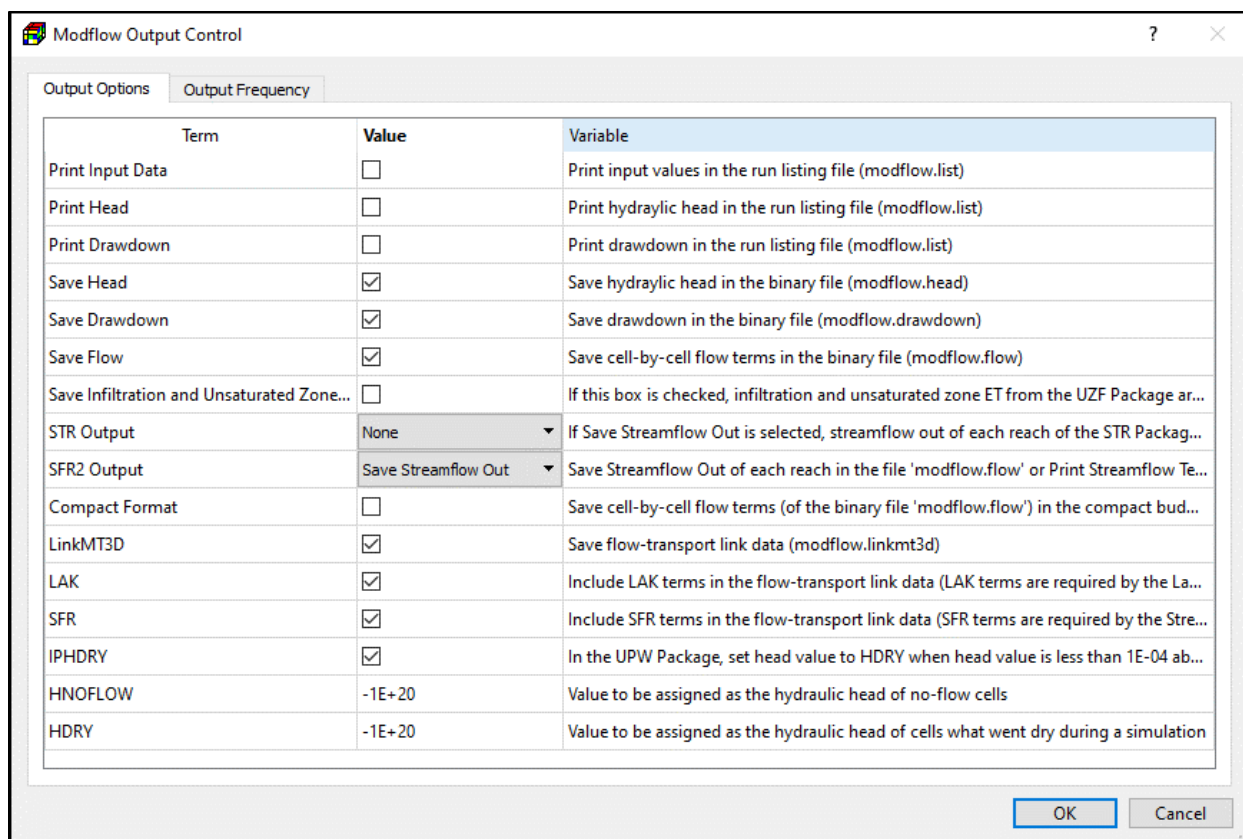


Figure 111. The Modflow Output Control dialog box

1. Output Options Tab

- **Print Input Data:** Check this box to write input values to the run listing file modflow.list.
- **Print Head:** Hydraulic heads in individual model cells are the primary result of a MODFLOW simulation. Check this box to write the calculated hydraulic head values to the run listing file modflow.list.
- **Print Drawdown:** Check this box to write the calculated drawdown values to the run listing file modflow.list.
- **Save Head:** Check this box to save the calculated hydraulic head values in the binary file modflow.head.
- **Save Drawdown:** Drawdown values in individual model cells are the differences between the initial hydraulic heads and the calculated hydraulic heads. If this box is checked, drawdown values are saved in the binary file modflow.drawdown.
- **Save Flow:** If this box is checked, cell-by-cell flow terms are saved in the binary file modflow.flow. Cell-by-cell flow terms include stress flows (e.g., pumping), storage terms, constant-head flow terms, leakage (from/to river, drain, stream reaches, etc.), and flows across cell faces for individual cells.

- **Save Infiltration and Unsaturated Zone ET:** If this box is checked, infiltration and unsaturated zone ET from the UZF Package are saved in the cell-by-cell flow file (modflow.flow.)
- **STR Output:**
 - If Save Streamflow Out is selected, streamflow out of each reach of the STR Package will be saved in the binary file 'modflow.flow'.
 - If None is selected, streamflow out will not be saved.
- **SFR2 Output:**
 - If “Save Streamflow Out” is selected, streamflow out of each reach will be saved in the file 'modflow.flow'.
 - If “Print Streamflow Terms” is selected, streamflow terms of each reach will be printed in the text file 'modflow.sfr2.list'. Streamflow terms include inflows, outflows, stream depth, width, streambed conductance, and head difference and gradient across the streambed.
 - If None is selected, neither streamflow out nor streamflow terms will be produced.
- **Compact Format:** Check this box to save the cell-by-cell flow terms in a compact format. ZoneBudget may not work correctly if SFR2 Output (see above) is set to "Save Streamflow Out" and this box is checked. Other existing software that work with the regular (non-compact) format may not be compatible with Compact Format.
- **LinkMT3D:** if this box is checked, the flow-transport link data will be saved in the binary file modflow.linkmt3d. This file is required by MT3D.
- **LAK⁵¹:** If this box is checked, the flow terms associated with the LAK Package will be saved to the binary file modflow.linkmt3d. This option is required by the Lake Transport (LKT) Package of MT3D-USGS. If the LKT Package is not used, you may check or uncheck the *LAK* box.
- **SFR⁵¹:** if this box is checked, the flow terms associated with the SFR package (such as in-stream flow) will be saved to the binary file modflow.linkmt3d. This option is required by the Streamflow Transport (SFT) Package of MT3D-USGS. If the SFT Package is not used, you may check or uncheck the *SFR* box.
- **IPHDRY:** If the UPW Package (of MODFLOW-NWT) is used, activating this option will set the head value to HDRY when head have is less than 1E-04 above the cell bottom. MODPATH works with MODFLOW-NWT provided that the IPHDRY is checked.
- **HNOFLOW:** This is the value to be assigned as the hydraulic head of no-flow cells.
- **HDRY:** This is the value to be assigned as the hydraulic head of cells what went dry during a simulation.

2. Output Frequency Tab

⁵¹ The executable of MODFLOW-2005 will not work if the SFR2 Package is activated and the *SFR* and/or *LAK* box is checked. If you need to use the SFT and/or the LKT Package, the respective *SFR* and/or *LAK* box must be checked, and you must set MODFLOW Version to either MODFLOW-NWT or MODFLOW-OWHM.

- **Stress Period and Time Step:** The simulation results of a time step are saved, if the stress period and the time step are even multiple of the respective specified value. If you intend to run a transport model, both values for Stress Period and Time Step should be set to 1.
- **Save the Result of the First Time Step of Each Stress Period:** if this box is checked, the results of the first time-step of each stress period are saved regardless of the specified values for Stress Period and Time Step.
- **Save the Result of the Last Time Step of Each Stress Period:** if this box is checked, the results of the last time step of each stress period are saved regardless of the specified values for Stress Period and Time Step.

3.9.1.4. Inspect Model Data

Select this menu item to check the geometry of the model and the consistency of the model data. Table 2 shows a list of items that are inspected by PM. The inspection report is saved in the *modflow_inspection_log.txt* file.

Table 2. Model data inspected by PM


Inspected Items	Criteria
Initial head at constant-head cells	Initial head may not be lower than the cell bottom elevation.
Layer thickness	Thickness may not be zero or negative.
Top and bottom elevation of layers	Model layers may not overlap each other.
Horizontal hydraulic conductivity	Must be present and greater than zero if it is required.
Horizontal anisotropy	Must be present and greater than zero if it is required.
Vertical hydraulic conductivity	Must be present and greater than zero if it is required.
Vertical leakance	Must be present and greater than zero if it is required.
Vertical anisotropy	Must be present and greater than zero if it is required.
Specific storage	Must be present and greater than or equal to zero if it is required.
Specific yield	Must be present and greater than or equal to zero if it is required.
Storage coefficient	Must be present and greater than or equal to zero if it is required.
Transmissivity	Must be present and greater than zero if it is required.
Wetting threshold	Must be present if it is required.
Drain Package Drain with Return Flow Package	Drain conductance must be greater than zero. Drain cell must not be a constant-head cell. Drain cell should not be a no-flow cell. Drain elevation should not be lower than cell bottom elevation.
Flow and Head Boundary Package	Specified-flow cell should not be a constant-head cell. Specified-flow cell should not be a no-flow cell. Specified-head cell should not be a constant-head cell. Specified-head cell should not be a no-flow cell.
General-head boundary Package	GHB conductance must be greater than zero. GHB cell should not be a constant-head cell. GHB cell should not be a no-flow cell. GHB head should not be lower than the cell bottom elevation.
Lake Package	Lake cell must be a no-flow cell. Lake cell may not be a re-wettable cell. Lake cell may not be overlain by non-lake cells.

Inspected Items	Criteria
	Lakebed leakance must be greater than or equal to zero. Lake number should not be less than zero or greater than the total number of lakes.
Multi-Node Well 2 Package	Well should not be in a no-flow cell. For nodes of type Screen Interval, the screen intervals must not overlap. The bottom of each screen must lie below its top, and the top must lie above the model's bottom boundary.
Reservoir Package	Reservoir bed vertical hydraulic conductivity may not be negative. Reservoir bed thickness must be greater than zero.
River Package	Riverbed conductance must be greater than zero. RIV cell should not be a constant-head cell. RIV cell should not be a no-flow cell. River stage should not be lower than the riverbed bottom elevation. River stage should not be lower than the cell bottom elevation. Riverbed bottom elevation should not be lower than the cell bottom elevation.
Stream (STR) Package	Reach numbers within a segment must be consecutive. Streambed conductance of a STR cell must be greater than zero. STR cell must not be a constant-head cell. STR cell must not be a no-flow cell. Stream stage should not be lower than the streambed bottom elevation.
Streamflow-Routing (SFR2) Package	Reach numbers within a segment must be consecutive. SFR2 cell must not be a constant-head cell. SFR2 cell must not be a no-flow cell. Reach length of a SFR2 cell should be greater than zero. The following values are checked only if they are used in the simulation: Manning's roughness coefficient of a segment must be greater than zero. Depth1 and Depth2 of a segment must be greater than zero. Width1 and Width2 of a segment must be greater than zero. Streambed conductivity of a SFR2 cell must be greater than zero.
Time-variant specified head Package	CHD cell should not be a no-flow cell.
Well Package	WEL cell must not be a constant-head cell. WEL cell should not be a no-flow cell.

3.9.1.5. Run

Select this menu item to open the Run Modflow dialog box (Figure 112). The dialog box can be used to start the flow simulation with MODFLOW, to build input files to MODFLOW, or to check the model data. The available settings of the dialog box are described below.

1. Table

- **The build box** . Prior to running a flow simulation, PM uses the user-specified data to build input files for MODFLOW. An input file will be built if its build box is checked. If an input file does not exist or the input data has been modified since the last build, its build box will be checked automatically. The input files will be saved in the Path displayed in the dialog box.

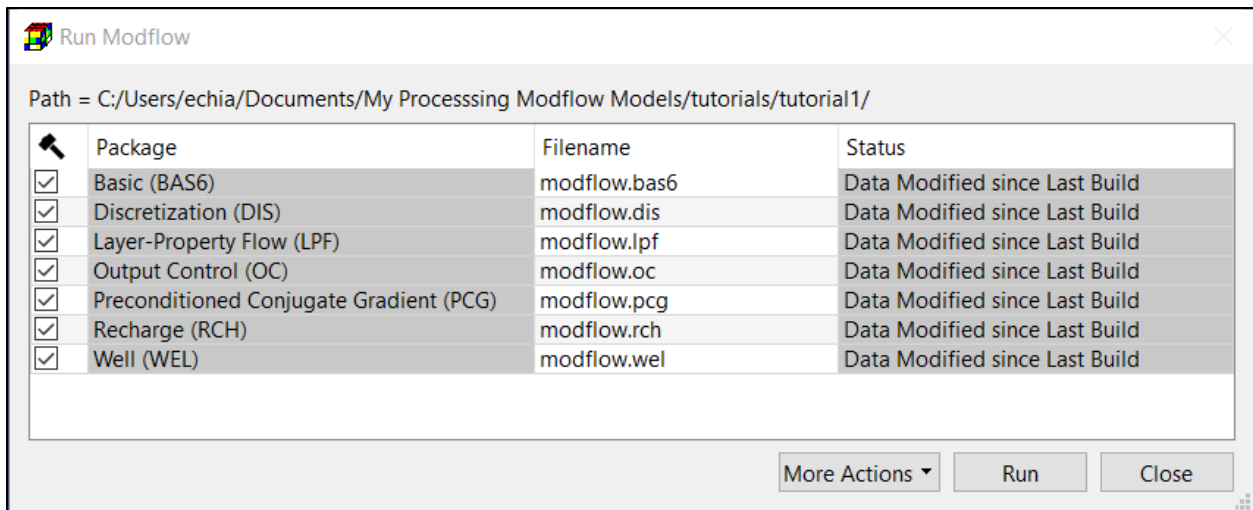


Figure 112. The Run Modflow dialog box

- **Package** is the name of the package of the flow model.
- **Filename** is the name of the input file.
- **Status** shows the status (such as Ready, Not Exist, etc.) of the input file.

2. More Actions

- **Inspect Model Data:** Select this action to check the geometry of the model and the consistency of the model data. Table 2 shows a list of items that are inspected by PM. The inspection report is saved in the *modflow_inspection_log.txt* file.
- **Build Input Files:** Select this action to build the input files with the checked build boxes. The name file modflow.nam and input file modflow.lmt for LMT8 package will also be built. The build log is stored in the *modflow_build_log.txt* file located in the model data folder.
- **Rebuild All Input Files:** Select this action to rebuild all input files, except for existing read-only input files. The build log is stored in the *modflow_build_log.txt* file.

3. **Run:** Click this button to build input files with the checked build boxes and the start the flow simulation. The flow simulation will be carried out by the executable file associated with the selected MODFLOW Version⁵². The MODFLOW run progress will be displayed in the MODFLOW Output window. You may terminate the simulation at any time by clicking on the Terminate button

⁵² See Section 1.3 for how to select a MODFLOW version. See Section 3.5.12 for how to assign executable files to various simulation codes

3.9.1.6. View

3.9.1.6.1. Run Listing File

Select this menu item to use the Text Editor (see Section 3.5.12) to display the run listing file `modflow.list`.

MODFLOW calculates a volumetric water budget for the entire model at the end of each time-step and save it in the run listing file. The volumetric water budget provides an indication of the overall acceptability of the numerical solution. In numerical solution techniques, the system of equations solved by a model consists of a flow continuity statement for each model cell. Continuity should therefore also exist for the total flows into and out of the entire model or a sub-region. This means that the difference between total inflow and total outflow should equal the total change in storage. The run listing file also contains other essential information. In case of difficulties in obtaining satisfactory model results, this supplementary information could be very helpful.

3.9.1.6.2. Model Inspection File



Select this menu item to use the Text Editor (see Section 3.5.12) to display the model inspection report file `modflow_check_log.txt`. See Section 3.9.1.5 and Table 2 for details about the model data that may be inspected by PM.

3.9.1.6.3. Scatter Chart

Select a respective item from the Scatter menu to display the scatter chart based on the head, drawdown, streamflow, stage, compaction, subsidence, or concentration data. For example, select *Scatter Chart > Head* to display the scatter chart based on the head data. The Scatter Chart tool is shown in Figure 113.

- 1. Chart:** The chart displays a scatter plot of simulated versus observed data (see the Data tab below). The x-axis represents the observed values, while the y-axis shows the calculated (simulated) values. For each observation point with more than one data entry, a linear least squares regression line can be displayed. This line models the relationship between observed and calculated values by fitting the linear equation $y = m \cdot x + b$, where m is the slope and b is the y-intercept. Scatter plots are commonly used to assess the goodness of fit between observed and simulated data. In the case of perfect agreement, all points fall on a 45° line, with a slope (m) of 1 and a coefficient of determination (r^2) equal to 1. For each observation point, the corresponding values of m , b , and r^2 can be shown in the chart legend. Additionally, the overall r^2 value—calculated from all visible data points—is displayed in the Observation Points tab (see below). Note: When only two data points are available, r^2 will always equal 1, as the regression line passes through both points exactly.

To interact with the chart, follow these steps:

- When the mouse cursor is moved into the chart, it becomes a crosshair (+), and the approximate observed and calculated values are displayed next to the cursor.
- **To highlight a regression line**, click on it. The corresponding row in the Observation Points table will be highlighted. Alternatively, you can highlight the regression of an observation point by clicking on the respective Name field in the Observation Points table.
- **To pan around the chart**, press and hold the left mouse button and then move the mouse.
- **To zoom in/out on any part of the chart**, place the mouse cursor within the chart, scroll the mouse wheel forward or backward.
- **Popup Menu:** You can right-click on a chart to display the popup menu. The items in the popup menu are described below.
 -  **Show Legend** or **Hide Legend:** This will show or hide the legend, which is displayed on the upper-right corner of the chart.
 -  **Full Extent:** This will cause the display of the chart to zoom out to its data extent.

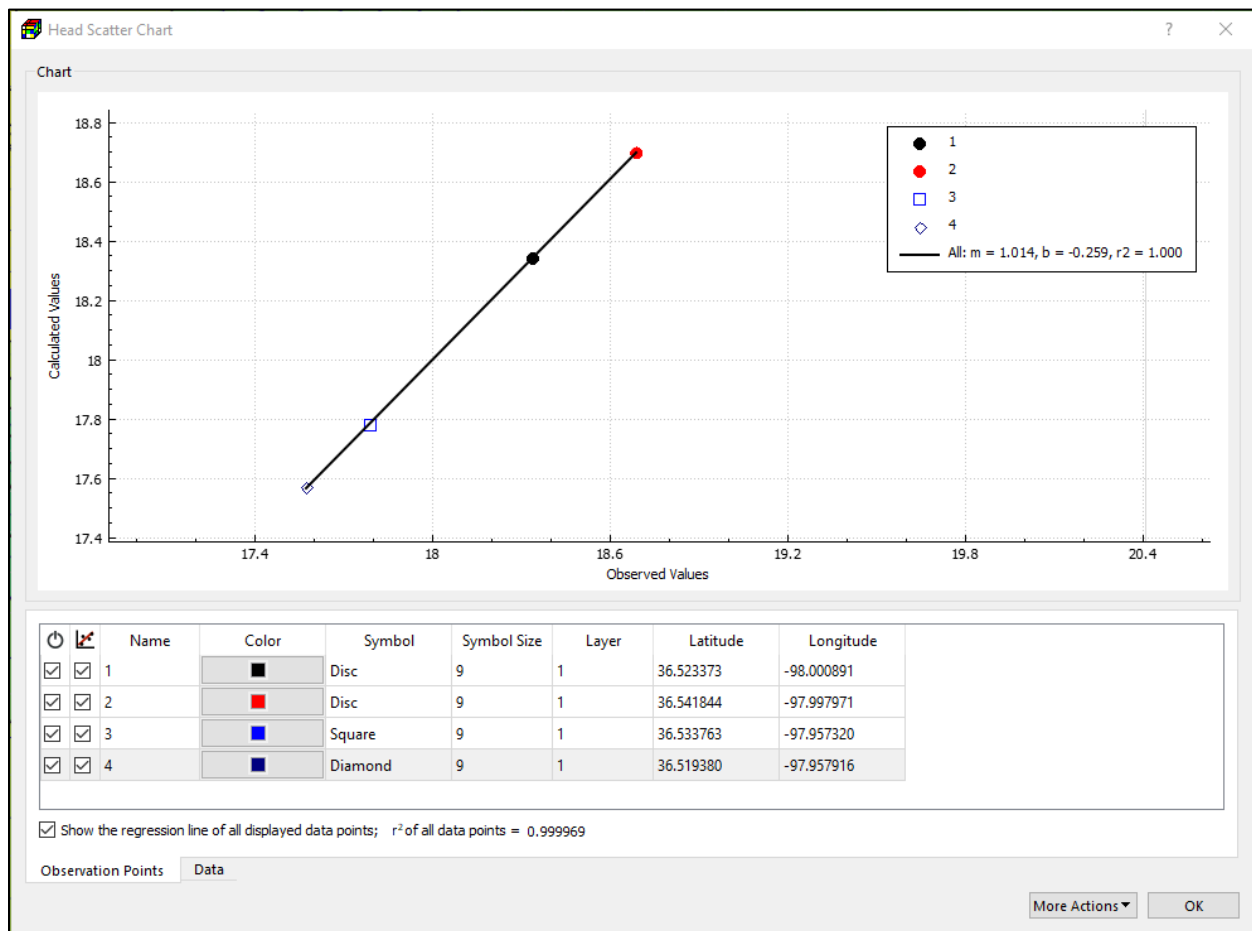




Figure 113. The Scatter Chart tool for hydraulic head

2. Observation Points Tab: This tab contains the observation points and their details. Each row in the table corresponds to an observation point. Observation points and observed values are defined by using the Observation Tool (Section 3.6.3).

- : Check/uncheck the box to turn on/off the display of the results in the chart. You may right-click on a row, and then select **Turn Off All But This** from a popup menu to switch off all other points.
- : Check/uncheck the box to turn on/off the display of the regression line based on the data of the respective observation point.
- **Name** (read-only) displays the name of the observation point.
- **Label** (read-only) displays the label of the observation point.
- **Color** defines the color for the display of the symbol and regression line in the chart. Click on the **Color** header to assign a color to all points.
- **Symbol** defines the symbol for data points in the chart.
- **Symbol Size** defines the size of the symbol for data points in the chart.
- **Layer** (read-only) is the model layer where the observation point is located.
- **Latitude** and **Longitude** (read-only) are the coordinates of the observation point.
- **Display regression line of all data points:** Check/uncheck this box to turn on/off the display of the regression line based on the data all displayed data points.
- **r² of all data points:** This is the coefficient of determination based on all displayed data points.

3. Data@SimTime Tab: This table contains a list of simulated values at model output times and the observed values (if available) interpolated to those times. Only the values of the selected observation point on the Observation Points tab are displayed. The columns of this table are as follows.

- **Observation Point** contains the name of the highlighted observation point.
- **Simulation Time** contains the model output times at which the simulated values are saved.
- **Date/Time** contains the calendar date/time that corresponds to the Simulation Time. The format of Date/Time corresponds to the ISO 8601 extended specification for representations of dates and times, taking the form YYYY-MM-DDTHH:mm:ssZ. Date/Time is only valid for years in the range 0 to 9999. Date/Time is blank for years beyond this range.
- **Observed Value** contains the observed values at the respective observation points and simulation times. The observed value at a given simulation time is obtained by linear interpolating user-specified observed values to that simulation time. If a simulation time lies outside of the span of the observed times, the corresponding observed value cannot be obtained by interpolation and remains blank. The average and variance of all observed values with valid calculated counterparts are given at the bottom of this table.
- **Calculated Value** contains the simulated values at the respective observation points and simulation times. For a given simulation time t (at which the hydraulic heads or

drawdown are saved in the binary file modflow.head or modflow.drawdown), the simulated value h_t at an observation point in a model layer is obtained by using the following equation:

$$h_{t,k} = \frac{\sum_{i=1}^4 (h_{i,t,k} \cdot A_i)}{\sum_{i=1}^4 A_i} \quad (43)$$

where

A_i are the areas as shown in Figure 114; for inactive cells $A_i = 0$

$h_{i,t,k}$ are the simulated head or drawdown values in layer k at the nodes surrounding the observation point for the simulation time t . For multi-layer observations, Equation (3) on page 67 is used to calculate the layer-weighted head or drawdown values. The average and variance of all pairs of calculated and observed values are given at the bottom of this table.

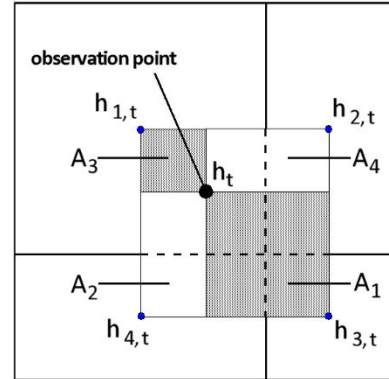


Figure 114. Interpolation of simulated head values to an observation point

- **Residual:** Residual is calculated as [Calculated Value - Observed Value] if both values are defined and valid, otherwise the value is blank. The average, variance, root mean squared error (RMSE), and coefficient of determination (R^2) of all valid residual values are given at the bottom of this table.

4. Data@ObsTime Tab: This table contains a list observation times, observed values, and the simulated values interpolated to the observation times. Only the values of the highlighted observation point on the Observation Points tab are displayed. The columns of this table are as follows.

- **Observation Point** contains the name of the highlighted observation point.
- **Observation Time** contains the observation times expressed as the simulation time (i.e., elapsed time since the start of the simulation).
- **Date/Time** contains the calendar date/time that corresponds to the Observation Time. The format of Date/Time corresponds to the ISO 8601 extended specification for representations of dates and times, taking the form YYYY-MM-DDTHH:mm:ssZ. Date/Time is only valid for years in the range 0 to 9999. Date/Time is blank for years beyond this range.
- **Observed Value** contains the observed values at the respective observation points and observation times. The average and variance of all observed values with valid calculated counterparts are given at the bottom of this table.
- **Calculated Value** contains the simulated head values at the respective observation points and observation times. The calculated value at a given observation time is obtained by linear interpolating the results of Equation (43) to that observation time. If an observation time lies outside of the simulation time span, the corresponding calculated value cannot be obtained by interpolation and remains blank. The average and variance of all pairs of calculated and observed values are given at the bottom of this table.

- **Residual:** Residual is calculated as [Calculated Value - Observed Value] if both values are defined and valid, otherwise the value is blank. The average, variance, root mean squared error (RMSE), and coefficient of determination (R^2) of all valid residual values are given at the bottom of this table.

5. More Actions

- **Activate:** Switch on the selected observation points⁵³.
- **Deactivate:** Switch off the selected observation points⁵³.
- **Save Chart:** Select this item to display the Save Image dialog box that allows you to save the chart as in a support image format (JPEG, PNG, or BMP) or in the PDF format.
- **Save Data:** Select this item to display the Save Data dialog box that allows you to save the content of the Data table in the following file types
 - **Comma Delimited [Data@SimTime] (*.csv):** This file contains a list of the simulated values at model output times and the observed values (if available) interpolated to those times⁵⁴. The first row of this file contains the column headers. The remaining rows contain the data. The columns are as follows.
 - ObservationPoint: Name of observation points
 - Simulation Time: Elapsed time since the start of the simulation, expressed in the model time unit.
 - DateTime: calendar date/time corresponding to simulation times.
 - ObservedValue: observed values (if available) interpolated to the simulation times.
 - CalculatedValue: calculated value at the simulation times.
 - Residual: Residual = CalculatedValue – ObservedValue
 - Layer(s): model layer(s) of the observation points
 - Weight(s): layer weights of the observation points
 - **Comma Delimited [Data@ObsTime] (*.csv):** This file contains a list observation times, observed values, and the simulated values interpolated to the observation times. The first row of this file contains the column headers. The remaining rows contain the data. The columns are the same as the “Comma Delimited [Data@SimTime] (*.csv)” file above.
 - **Excel Workbook [All Values] (*.xlsx):** This workbook contains four worksheets.
 - The first worksheet, *ObservationPoints*, contains the observation points. Observation points with Target = None in the Observation Tool (see Section 3.6.3) are not included.
 - The *CalculatedValues* worksheet contains two columns for each observation point. The first column contains simulation date/times, the second column contains calculated values at the given simulation date/times.

⁵³ You may select multiple rows from the Observation Point table by using combinations of Ctrl+click and Shift+click.

⁵⁴ Use “Excel Workbook [Data@SimTime] (*.xlsx)” or “Excel Workbook [Obs. vs. Cal.] (*.xlsx)” to export pairs of simulated and observed values.

- The *CalibrationTargets* worksheet contains two columns for each observation point. The first column contains observation date/times of selected calibration targets, the second column contains calibration targets at the given observation date/times.
- The *ObservedValues* worksheet contains two columns for each observation point. The first column contains all observation date/times of an observation point, the second column contains observed values at the given observation date/times.
- **Excel Workbook [Data@SimTime] (*.xlsx):** This workbook contains several worksheets.
 - The first worksheet, *ObservationPoints*, contains a list of the observation points that have at least a valid calculated value.
 - A worksheet is added to the Excel workbook for each of the observation points listed in the first worksheet. The names of the observation points are used as the worksheet names. Each worksheet contains the *simulation* times and pairs of calculated and observation values at those simulation times. The observed value at a given simulation time is obtained by linear interpolating the user-specified observed values to that simulation time. Therefore, a row is included only if a simulation date/time lies within the observation time span.
 - Statistics of the calculated and observation value pairs are given in the *ObservationPoints* worksheet and at the end of each worksheet.
- **Excel Workbook [Data@ObsTime] (*.xlsx):** This workbook contains several worksheets.
 - The first worksheet, *ObservationPoints*, contains a list of the observation points that have at least one valid observed value, and the statistics of the calculated and observation value pairs.
 - A worksheet is added to the Excel workbook for each of the observation points listed in the first worksheet. The names of the observation points are used as the worksheet names. Each worksheet contains the *observation* times and pairs of observed and calculated values at those observation times. The calculated values are linearly interpolated to the observation times. Therefore, a row is included only if an observation date/time lies within the simulation time span.
 - Statistics of the calculated and observation value pairs are given in the *ObservationPoints* worksheet and at the end of each worksheet.
- **Excel Workbook [Obs. vs. Cal.] (*.xlsx):** This workbook contains five worksheets.
 - The *ObservationPoints* worksheet contains a list of the observation points that have at least a pair of valid observed and calculated data.
 - The *Data1@SimTime* worksheet contains three columns for each observation point. The first column contains simulation date/times; the second column contains calculated values at the given simulation date/times; and the last column contains the observed values interpolated to the given simulation

date/time. A row is included only if an observation value can be interpolated to the given simulation date/time.



- The *Data2@SimTime* worksheet contains four columns. The first column contains Point Name; the second column contains simulation date/times; the third column contains calculated values at the given simulation date/times; and the last column contains the observed values interpolated to the given simulation date/time. A row is included only if an observation value can be interpolated to the given simulation date/time.
 - The *Data1@ObsTime* worksheet contains three columns for each observation point. The first column contains observation date/time; the second column contains calculated values interpolated to the given observation date/time; and the last column contains the observed values at given observation date/time. A row is included only if a calculated value can be interpolated to the given observation date/time.
 - The *Data2@ObsTime* worksheet contains four columns. first column contains Point Name; the second column contains observation date/time; the third column contains calculated values interpolated to the given observation date/time; and the last column contains the observed values at given observation date/time. A row is included only if a calculated value can be interpolated to the given observation date/time.
- **Print Chart:** Select this item to open the Print dialog box that enables you to print the chart.

3.9.1.6.4. Line Chart

The Line Chart tool can be used to display time series of simulated and observed head, drawdown, streamflow, lake stage, compaction, and subsidence⁵⁵. For example, to display the line chart based on the head data, select *Line Chart > Head*. The Line Chart tool is shown in Figure 115.

1. **Chart:** The chart displays a line graph using the simulated and observed data (see Data tab below). The x-axis represents the simulation time and the y-axis represents the calculated and observed data. Two time-series curves are displayed for each observation point. Calculated and observed data points are represented by graphic symbols. Solid lines connect the calculated data and dashed lines connect the observed data. To interact with the chart, follow these steps:
 - When the mouse cursor is moved into the chart, it becomes a crosshair (+), and the head (or drawdown) value, simulation time, and corresponding calendar date/time are displayed next to the cursor.
 - **To pan around the chart**, move the mouse cursor into the chart, press and hold the left mouse button, and then move the mouse.

⁵⁵ Compaction and subsidence are simulated by the IBS or SUB package. The simulated subsidence values are associated with the first model layer.

- **To zoom in/out on any part of the chart**, place the mouse cursor within the chart, scroll the mouse wheel forward or backward.
- **You can limit the zoom and pan actions** to the x or y direction by selecting the x or y-axis first. An axis is selected by clicking on it. A selected axis is colored in blue. To deselect an axis, simply click on any part of the chart.
- **To highlight the time-series curves of an observation point**, click on a curve or a data point symbol. The corresponding row in the Observation Points table will be highlighted. Alternatively, you can highlight the time-series curves of an observation point by clicking on the respective Name field in the Observation Points table.
- **Popup Menu:** You can right-click on a chart to display the popup menu. The items in the popup menu are described below.
 -  **Full Extent:** This will cause the display of the chart to zoom out to its data extent.
 -  **Show Legend** or **Hide Legend:** This will show or hide the legend, which is displayed on the upper-right corner of the chart.
 - **Use Logarithmic Scale** or **Use Linear Scale:** This allows you to switch the y-axis between logarithmic and linear scales. This option is only available when all calculated and observed values are greater than zero.

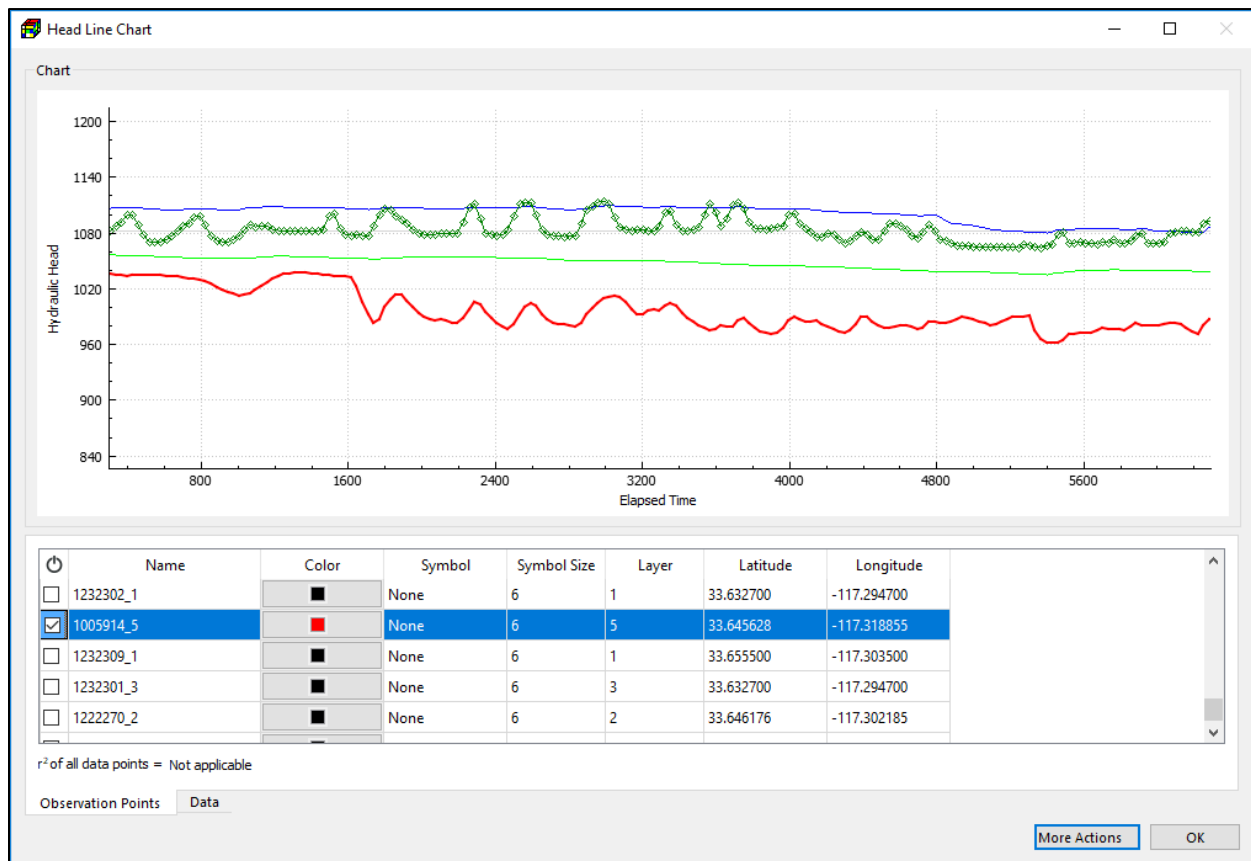



Figure 115. The Line Chart tool for hydraulic heads

2. **Observation Points Tab:** This tab contains the observation points and their details. Each row in the table corresponds to an observation point. Observation points and observed values are defined by using the Observation Tool (Section 3.6.3).
 - : Check/uncheck the box to turn on/off the display of the results in the chart.
 - **Name** (read-only) displays the name of the observation point.
 - **Label** (read-only) displays the label of the observation point.
 - **Color** defines the color for the display of the symbol and time-series curves in the chart.
 - **Symbol** defines the symbol for data points in the chart.
 - **Symbol Size** defines the size of the symbol for data points in the chart.
 - **Layer** (read-only) is the model layer where the observation point is located.
 - **Latitude** and **Longitude** (read-only) are the coordinates of the observation point.
 - **r²**: This is the coefficient of determination based on the calculated and observed data pairs of the observation points that are turned on. Note that r^2 always equals to 1 if there are only two data points.
3. **Data@SimTime Tab:** This table contains a list of simulated values at model output times and the observed values (if available) interpolated to those times. The contents of this table are the same as those of the Scatter Chart dialog box. See the previous section for details.
4. **Data@ObsTime Tab:** This table contains a list observation times, observed values, and the simulated values interpolated to the observation times. The contents of this table are the same as those of the Scatter Chart dialog box. See the previous section for details.
5. **More Actions:** The functionalities of this menu are the same as those of the Scatter Chart dialog box. See the previous section for details.

3.9.1.6.5. Volume Summary

This menu is intended to be used for the calculation of the groundwater volume based on the starting hydraulic head, the simulated hydraulic head⁵⁶ (stored in the modflow.head file), layer types (unconfined, confined, or convertible), layer geometry (top and bottom elevation of layers), and aquifer parameters (S, Ss, and Sy). If aquifer parameters are not specified, their respective default values are used for the calculation. Dry or no-flow cells are excluded from the volume calculation.

Zones

Select *Volume Summary > Zones...* to define zone numbers with the Data Editor. The groundwater volume is calculated for each defined zone. If zone numbers are not defined, the entire model will be treated as a single zone, with the default zone number set to 1. Zone numbers from 0 to

⁵⁶ If the simulated saturated thickness of a cell (in a unconfined or convertible layer) is less than the Dry Cell Tolerance value defined in the Preferences dialog box, the cell is excluded from the storage calculation.

999 are allowed. Cells with zone number 0 are not associated with any zone and are excluded from the calculation.

Run

Select *Volume Summary > Run...* to calculate the groundwater volume. The calculation results are stored in a comma-separated value file. Each record in the file represents storage terms in each zone, time step, and layer. A record includes the following terms.

- **Zone** is the zone number.
- **SimulationTime** is the simulation time.
- **Period** is the stress period. The records with Period = 0 represent the initial condition based on the starting hydraulic head.
- **TimeStep** is the time step within the given Period.
- **Date** is the calendar date in the form of year-month-day. The date corresponds to the given Period and TimeStep.
- **Layer** is the model layer.
- **Vsy** [L³] is the groundwater volume stored in the pore space (based on Sy) of the given layer and in the given date. The groundwater volume stored in the pore space of a cell (*vsy*) is calculated based on the following rules.

- For confined/convertible layers:

$$\begin{aligned} vsy &= (t - b) \times Area \times S_y && \text{if } h \geq t \\ vsy &= (h - b) \times Area \times S_y && \text{if } b < h < t \\ vsy &= 0 && \text{if } h \leq b \end{aligned}$$

Where

S_y [-] is the specific yield of the cell.

- For unconfined layers:

$$\begin{aligned} vsy &= (h - b) \times Area \times S_y && \text{if } h > b \\ vsy &= 0 && \text{if } h \leq b \end{aligned}$$

- **Vs** [L³] is the groundwater volume stored due to compressibility (based on S or Ss) of the given layer and in the given date. The groundwater volume stored in a cell due to compressibility (*vs*) is calculated based on the following rules.

- For confined/convertible layers:

$$\begin{aligned} vs &= (t - b) \times Area \times S_s \times (h - z) && \text{if } h \geq t \\ vs &= (h - b) \times Area \times S_s \times \left(\frac{h - b}{2}\right) && \text{if } b < h < t \\ vs &= 0 && \text{if } h \leq b \end{aligned}$$

Where

S_s [L⁻¹] is the specific storage of the cell,

h [L] is the hydraulic head in the cell,

z [L] is the elevation of the center of the cell,

t [L] is the elevation of the top of the cell, and

b [L] is the elevation of the bottom of the cell, and
 $Area$ [L²] is the area of the cell,
 $(h - b)$ is the saturated thickness of the cell when it is not under confined condition, and
 $(h - b)/2$ represents the average hydraulic head that acts on the water within the cell when it is not under confined condition.

- For unconfined layers:

$$\begin{aligned}
 vs &= (h - b) \times Area \times S_s \times ((h - b)/2) && \text{if } h > b \\
 vs &= 0 && \text{if } h \leq b
 \end{aligned}$$

- **Sum** [L³] is the sum of V_{sy} and V_s .

3.9.1.6.6. Stream Gage Output

Select this menu item to use the Text Editor (see Section 3.5.12) to display the output file `modflow.streamgageoutput` of the Gage Package. That output file stores the simulated stream stage, outflow, as well as depth, width and flow at midpoint of stream reaches that are designated as gages. This menu item is enabled only if the `modflow.streamgageoutput` file exists.

3.9.1.6.7. Lake Gage Output

Select this menu item to use the Text Editor (see Section 3.5.12) to display the output file `modflow.lakegageoutput` of the Gage Package. That output file stores the simulated values of lake stage for lakes that are designated as gages. This menu item is enabled only if the `modflow.lakegageoutput` file exists.

3.9.2. ZoneBudget

In certain cases, it is beneficial to calculate flow terms for specific zones (subregions) within the model domain. To support such analyses, MODFLOW stores computed flow terms for individual cells in the **modflow.flow** file, also known as the cell-by-cell flow file. These flow terms include:

1. Inflows or outflows due to external stresses or boundaries (e.g., pumping wells, recharge, or general-head boundaries).
2. Changes in storage.
3. Flows to or from constant-head boundaries.
4. Flows across the faces of model cells.

ZoneBudget (Harbaugh, 1990; 2009) is a program designed to read these flow terms from the cell-by-cell flow file and generate water budgets for user-defined zones within the model domain. Zones are created by assigning zone numbers to the model cells. To compute water budgets with ZoneBudget, follow the steps outlined below. Before running ZoneBudget, ensure that its output files (i.e., `zonebudget.zblst`, `zonebudget.2.csv`, `zonebudget.csv` in the model folder) are not open in any other application.

1. Define Zone Numbers with the Data Editor

- Navigate to *Models > ZoneBudget > Zones* to define the zone numbers.
- If no zone numbers are defined, the entire model will be treated as a single zone, with the zone number set to 1.
- Zone numbers range from 0 to 999. A zone number of 0 indicates that a cell is not part of any zone and will be excluded from the zone budget calculation.

2. Run the ZoneBudget Program

- Select *Models > ZoneBudget > Run* to initiate the ZoneBudget program.
- PM will build the input file for ZoneBudget and launch the program.
- The progress of the ZoneBudget run will be displayed in the **ZoneBudget Output** window. You can terminate the program at any time by clicking the **Terminate** button. Once the calculation is complete, click the **Close** button to exit the window.

3. Review the Results

- Select *Models > ZoneBudget > View > Listing File* to open the **zonebudget.zblst** file, which provides the water budget for all zones at each time step where flow terms are saved in the cell-by-cell flow file.
- Select *Models > ZoneBudget > View > CSV File* to open the **zonebudget.csv** file, which logs flow terms for each time step where cell-by-cell flow data is recorded. Each row contains an inflow or outflow term for individual zones. Zones are arranged in columns, allowing for easy comparison of budget terms across all zones. The file is formatted for smooth integration into spreadsheet applications.
- Select *Models > ZoneBudget > View > CSV2 File*⁵⁷ to open the **zonebudget.2.csv** file, which provides a detailed water budget for a single zone at a specific time step in a single row. Each column corresponds to a distinct inflow or outflow term, facilitating in-depth analysis. Rows can be sorted by time within a spreadsheet application, making it easy to track changes in budget terms over time. The file is formatted for effortless import into spreadsheet software.

If you have multiple zone definitions, you can edit and save them as separate 3D MODFLOW Array files during Step 1. Once completed, navigate to *Models > ZoneBudget > Batch Run* and select the desired 3D MODFLOW Array files in the Open File dialog box. PM will execute the ZoneBudget program for each selected file, storing the results in dedicated subfolders within the model data folder. Each subfolder will be named after the corresponding file's base name.

3.9.3. MT3D/SEAWAT

This menu provides the interface to MT3D-USGS (Bedekar, et al., 2016) and SEAWAT (Langevin, et al., 2007). The former simulates transport processes based on the flow results of MODFLOW,

⁵⁷ As of this writing, if a package (such as general-head boundary) is active in the flow model but has no defined cells for a given stress period, the inflow and outflow budget terms for that package will be omitted from the CSV2 file for that period, potentially resulting in an incorrectly formatted file.

thus a MODFLOW run must be completed prior to using MT3D-USGS. SEAWAT couples the flow and transport processes, therefore does not require a separate MODFLOW run.

3.9.3.1. Settings

The first step to use MT3D or SEAWAT is to define the species, packages, and the reaction type in the Transport Settings dialog box (Figure 116). The features of this dialog box are detailed below.

1. More Actions

- **Import MT3D Model:** Select this option to import a transport model saved in the MT3D-USGS (Bedekar, et al., 2016) or MT3DMS (Zheng, et al., 1999) format. After selecting this option, you will need to (1) select the name file of the MODFLOW model, and then (2) select the name file of the MT3D model. The grid size of the selected MODFLOW and MT3D models must be identical. All packages of MT3D listed in Section 5.3.2.1 may be imported.
- **Import SEAWAT Model:** Select this option to import a transport model saved in the SEAWAT (Langevin, et al., 2007) format. All packages of MT3D and SEAWAT listed in Sections 5.3.2.1 and 5.3.4.1 may be imported.

2. **Model Tab** (Figure 116) defines the transport model and packages to be used in the simulation.

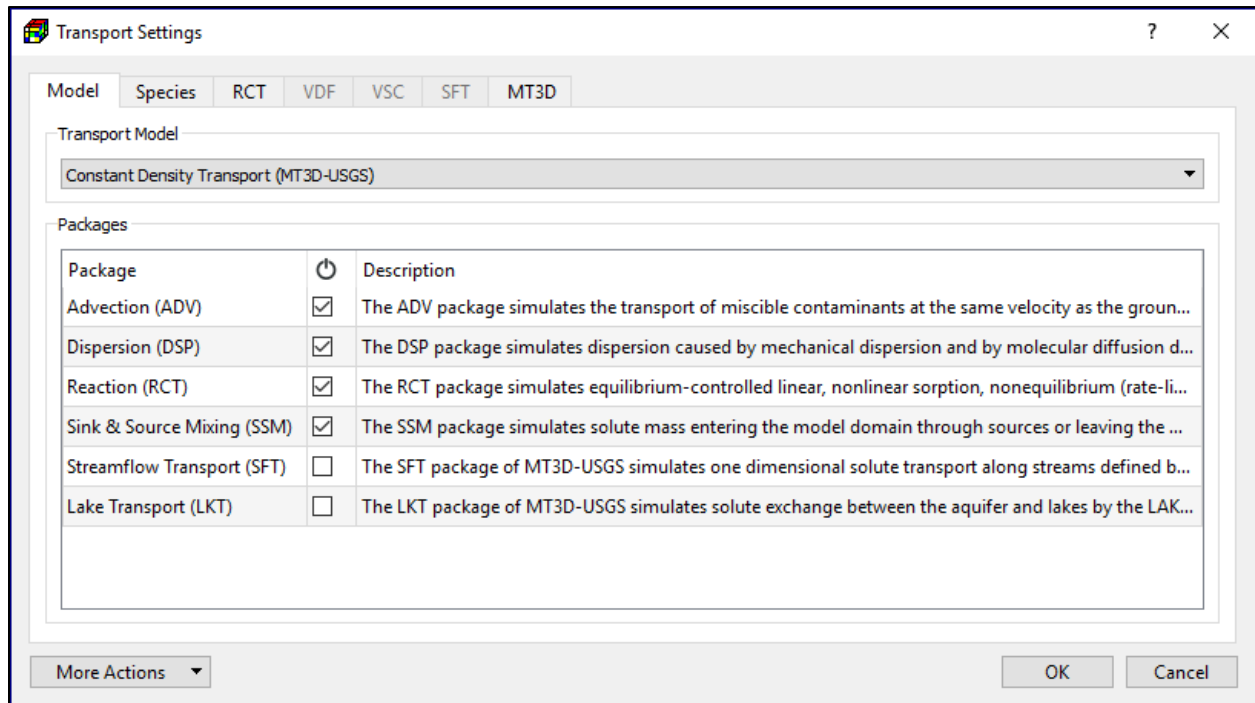



Figure 116. The Model tab of the Transport Settings dialog box

- **Transport Model** contains the following items:
 - **Constant Density Transport (MT3D-USGS):** If this item is selected, the transport simulation will be carried out by the executable file of MT3D-USGS defined in the Preferences dialog box (Figure 37). The transport process will be density independent.
 - **Constant Density Transport (MT3DMS):** If this item is selected, the transport simulation will be carried out by the executable file of MT3DMS defined in the Preferences dialog box (Figure 37). The transport process will be density independent. If your model uses any of the LAK3, SFR2, or UZF1 packages with MODFLOW-NWT or MODFLOW-OWHM, you should use MT3D-USGS instead of MT3DMS, because the latter is not compatible with the flow-transport link data file (modflow.linkmt3d) produced by MODFLOW-NWT and MODFLOW-OWHM when LAK3, SFR2, or UZF1 is present.
 - **Variable Density Flow and Transport (SEAWAT):** If this item is selected, the transport simulation will be carried out the executable file of SEAWAT defined in the Preferences dialog box (Figure 37). The transport process can be density dependent.
- **Packages:** The table contains the available packages of the selected transport model. Check/uncheck the boxes in the column to turn on/off packages.
 - **Advection (ADV):** The ADV package simulates the transport of miscible contaminants at the same velocity as the groundwater.
 - **Dispersion (DSP):** The DSP package simulates dispersion caused by mechanical dispersion and by molecular diffusion driven by concentration gradients.
 - **Reaction (RCT):** The RCT package simulates equilibrium-controlled linear, nonlinear sorption, nonequilibrium (rate-limited) sorption, and kinetic reactions.
 - **Sink & Source Mixing (SSM):** The SSM package simulates solute mass entering the model domain through sources or leaving the model domain through sinks.
 - **Streamflow Transport (SFT):** The SFT package of MT3D-USGS simulates one dimensional solute transport in the stream network that is defined by the SFR2 package. If this package is activated, it is required to use MODFLOW-NWT or MODFLOW-OWHM for the flow simulation^{48, 58}.
 - **Lake Transport (LKT):** The Lake Transport (LKT) Package calculates solute concentrations in lakes. The package uses simulated flows calculated by the Lake (LAK3) Package of MODFLOW. Solute entering the lake is instantaneously mixed throughout the entire volume of the lake while the constituent concentration in lake seepage is equal to the concentration in the lake.
 - **Variable-Density Flow (VDF):** The VDF Process of SEAWAT solves the variable-density ground-water flow equation. Activate this package to include the density effect in the simulation.

⁵⁸ The current version of MODFLOW-2005 is not compatible with the SFT Package.

- **Viscosity (VSC):** The VSC package of SEAWAT simulates effects of viscosity variations that are implemented in the LPF package⁵⁹.
3. **Species Tab** (Figure 117) defines the species to be used in the simulation. The table contains the species and related parameters.
- : Check/uncheck the boxes in this column to activate/deactivate species.
 - **Species Name:** Name of the species.
 - **Yield:** The value of the yield coefficient between the current species and the next species. This is used only if the first-order chain reaction is simulated.
 - **Density:** This is used only if the Variable Density Flow (VDF) process of SEAWAT is turned on. If the Density box of a species is checked, the concentration of that species will affect the fluid density (see DRHODC below). If all Density boxes are cleared, SEAWAT runs in the uncoupled mode and you must specify the fluid density (see Section 3.9.3.3.5) and has the option to determine the density of the source fluid using WELDENS, RIVDEN and GHBDENS in the VDF Tab.
 - **DRHODC** is the slope that relates fluid density (ρ) to solute concentration (C), i.e., $\partial\rho/\partial C$. This is used only if the Variable Density Flow (VDF) process of SEAWAT is turned on.
 - **CRHOREF** is the reference concentration (i.e., C_0) for the species. For most simulations, CRHOREF should be specified as zero. This is used only if the Variable Density Flow (VDF) process of SEAWAT is turned on.

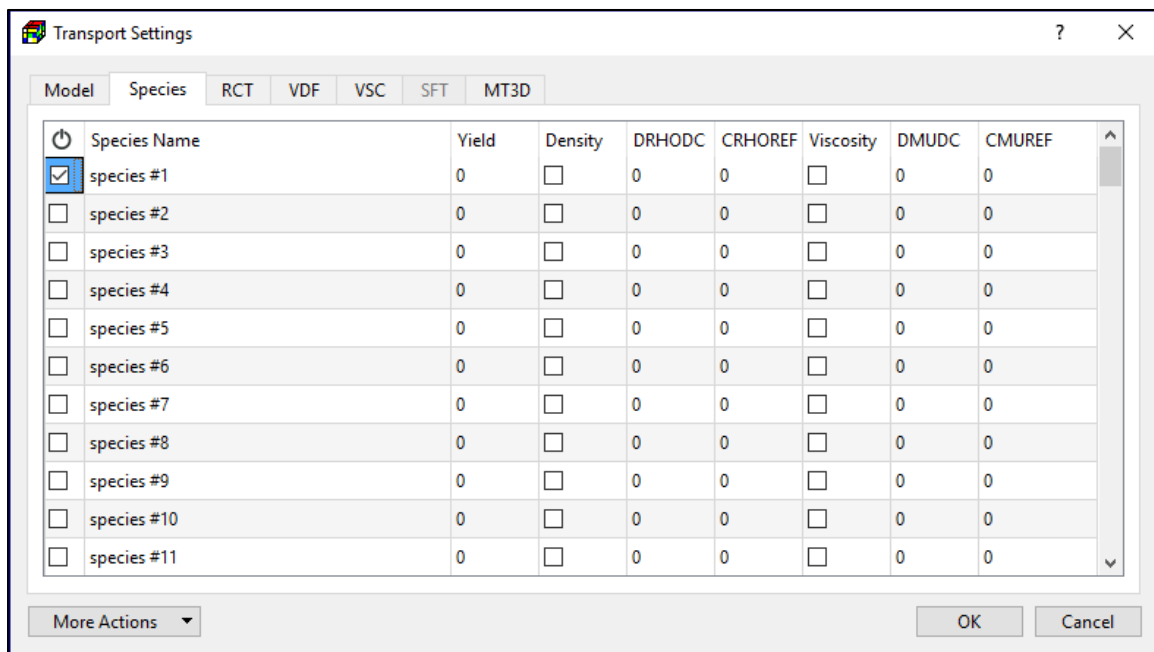


Figure 117. The Species Tab of the Transport Settings dialog box

⁵⁹ The effects of viscosity variations are implemented only for the Layer-Property Flow (LPF) Package; the Block-Centered Flow (BCF) and Upstream-Weighting (UPW) Packages cannot be used with the VSC Package.

- **Viscosity:** This is used only if the Viscosity (VSC) package of SEAWAT is turned on. If the Viscosity box of a species is checked, the concentration of that species will affect the fluid viscosity (see DMUDC below). If all Viscosity boxes are cleared, you must specify the viscosity value (see Section 3.9.3.3.6).
 - **DMUDC** is the slope that relates dynamic viscosity (μ) to solute concentration (C), i.e., $\partial\mu/\partial C$. This is used only if the Viscosity (VSC) package of SEAWAT is turned on.
 - **CMUREF** is the reference concentration for the species. This is used only if the Viscosity (VSC) package of SEAWAT is turned on.
4. **RCT Tab** (Figure 118) contains options for the Reaction Package. The available sorption and reaction types and their required parameters are described as follows. The required parameters for the selected options must be specified in the *Models > MT3D/SEAWAT > Packages > Reaction (RCT)* menu.
- **Sorption:**
 - **Sorption Type** defines the type of sorption or dual-domain transfer. Equilibrium-controlled sorption isotherms are implemented in MT3D through use of the retardation factor R .
 - **No sorption:** Sorption is not simulated.
 - **Linear Isotherm** simulates linear equilibrium sorption and assumes that the sorbed concentration \bar{C} is directly proportional to the dissolved concentration C . The retardation factor R is therefore independent of the concentration

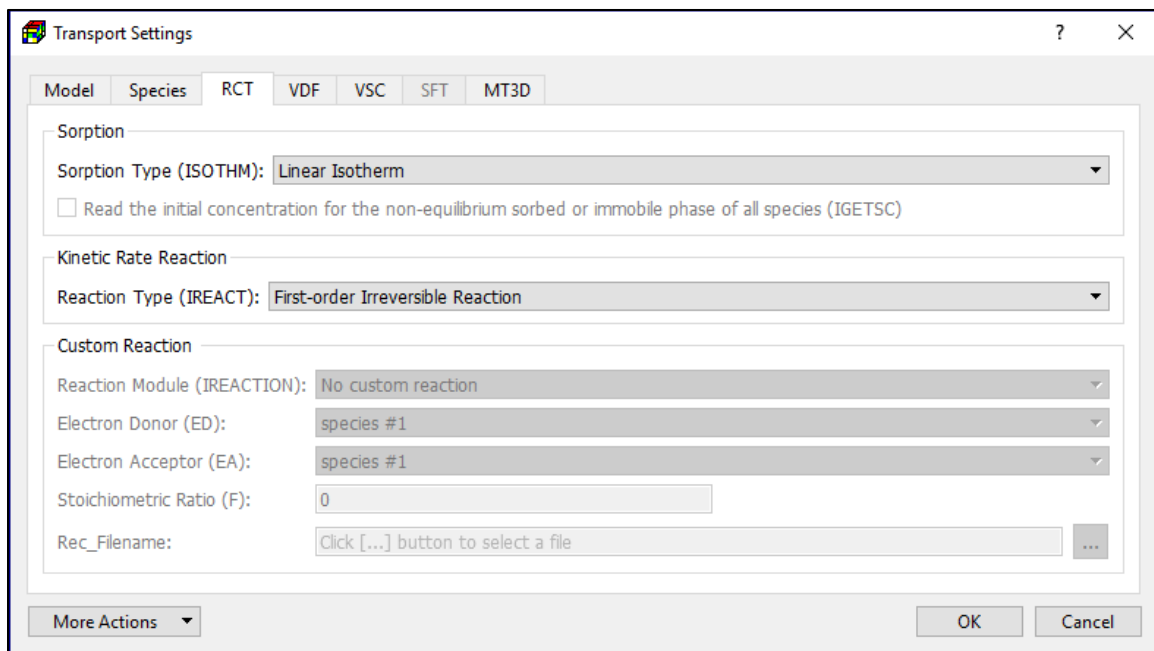


Figure 118. The RCT tab of the Transport Settings dialog box

values and is calculated only once for each cell at the beginning of the simulation.

$$\bar{C} = K_d \cdot C \quad (44)$$

$$R = 1 + \frac{\rho_b}{n} \cdot K_d \quad (45)$$

where n [-] is the porosity of the porous medium in a cell; K_d [L^3M^{-1}] is the distribution coefficient that depends on the solute species, nature of the porous medium, and other conditions of the system; and ρ_b [ML^{-3}] is the bulk density of the porous medium of the cell.

- **Freundlich Isotherm** simulates nonlinear equilibrium sorption with the following equation.

$$\bar{C} = K_f \cdot C^a \quad (46)$$

The retardation factor at the beginning of a transport step is calculated by

$$R = 1 + \frac{\rho_b}{n} \cdot a \cdot C^{a-1} \cdot K_f \quad (47)$$

Where C is the solute concentration in a cell at the beginning of a transport step; a [-] is the Freundlich exponent; and K_f [L^3M^{-1}] is the Freundlich constant.

- **Langmuir Isotherm** simulates nonlinear equilibrium sorption with the following equation.

$$\bar{C} = (K_L \cdot \bar{S} \cdot C) / (1 + K_L \cdot C) \quad (48)$$

The retardation factor at the beginning of a transport step is calculated by

$$R = 1 + \frac{\rho_b}{n} \cdot \frac{K_L \cdot \bar{S}}{(1 + K_L \cdot C)^2} \quad (49)$$

where K_L [L^3M^{-1}] is the Langmuir constant and \bar{S} [MM^{-1}] is the sorption capacity of the porous medium.

- **First-order Kinetic Sorption** simulates nonequilibrium sorption. It is assumed that sorption can be represented by a first-order reversible kinetic sorption defined by

$$\rho_b \cdot \frac{\partial \bar{C}}{\partial t} = \beta \cdot \left(C - \frac{\bar{C}}{K_d} \right) \quad (50)$$

Where β [T^{-1}] is the first-order mass transfer rate between the dissolved and sorbed phases.

- **Dual-Domain Mass Transfer (without Sorption), Dual-Domain Mass Transfer (with Sorption), and Dual-Domain Mass Transfer (with Different Sorption Coefficients in Mobile and Immobile Domains):** Dual-domain means that two kinds of continuum, e.g. a fractured medium and the porous medium, exist simultaneously in the same spatial region, i.e., the same model cells. In fractured aquifers, water moves faster along fractures than it does in a rock formation and the solute transport is often controlled by advection along the fractures and dominated by dispersion in the porous block along the fractures.

MT3D uses the dual-domain concept to approach extremely heterogeneous porous media or media composed of fractures and pores. In this approach, the effective porosity is used as the primary porosity for the pore spaces filled with mobile water (i.e., fractures) and the secondary porosity for the pore spaces filled with immobile water (i.e., rock formation) is defined separately. The sum of the primary and the secondary porosities is the total porosity of the medium.

The sorption process in the mobile and immobile domain is simulated by using the linear isotherm.

The exchange of the dissolved mass between the mobile and immobile domains can be defined by

$$n_{im} \cdot \frac{\partial C_{im}}{\partial t} = \zeta \cdot (C_m - C_{im}) \quad (51)$$

where n_{im} [-] is the porosity of the immobile domain; C_{im} [ML⁻³] is the solute concentration in the immobile domain; C_m [ML⁻³] is the solute concentration in the mobile domain; and ζ [T⁻¹] is the first-order mass transfer rate between the mobile and immobile domains. As the mass transfer rate ζ increases, the dual-domain model functions more and more like the single-domain model with a porosity approaching the total porosity of the porous medium.

- **IGETSC** is a flag indicating whether the initial concentration for the non-equilibrium sorbed or immobile phase of all species should be read when first-order kinetic sorption or dual-domain mass transfer is simulated.
 - If this box is checked, the initial concentration for the sorbed phase or immobile liquid phase of all species will be used in the simulation.
 - If this box is unchecked, the initial concentration for the sorbed or immobile phase is not used. The sorbed phase is assumed to be in equilibrium with the dissolved phase and the immobile domain is assumed to have zero concentration.
- **Kinetic Rate Reaction** defines the type of kinetic rate reaction. Note that *first-order irreversible reaction* and *MONOD kinetic reaction* are not intended for modeling chemical reactions between species.
 - **No Kinetic Rate Reaction:** kinetic rate reaction is not simulated.
 - **Zero-order Reaction:** The reaction rate of a zero-order reaction is constant and independent of the concentration of reacting substances. In MT3D, the zero-order reaction rate coefficients for the dissolved phase (in [ML⁻³T⁻¹]) and sorbed phase (in [MM⁻¹T⁻¹]) may be specified separately. The convention for both coefficients are positive values for decay and negative values for production.
 - **First-order Irreversible Reaction:** The concentration change due to the chemical reaction from one transport step to another at a cell can be expressed as

$$\Delta C = -\frac{\Delta t}{R} \cdot \left[\lambda_1 \cdot C + \lambda_2 \cdot \frac{\rho_b}{n} \cdot \bar{C} \right] \quad (52)$$

where λ_1 [T⁻¹] is the first-order rate for the dissolved phase; λ_2 [T⁻¹] is the first-order rate for the sorbed phase; Δt is the transport time-step; and \bar{C} is the mass of the solute species adsorbed on the solids per unit bulk dry mass of the porous medium at the beginning of each transport step. \bar{C} is in equilibrium with solute concentration C in the cell.

The rate constant λ is usually given in terms of the half-life. Generally, if the reaction is radioactive decay, λ_2 should be set equal to λ_1 . However, for certain types of biodegradation, λ_2 may be different.

- **First-order Chain Reaction:** The first-order chain reaction is implemented in MT3D for both dissolved and sorbed phases. Considering the dissolved phase, the changes in the concentration values of involved species within a transport time-step are calculated in the following sequential order:

$$\begin{aligned}\frac{\Delta C^1}{\Delta t} &= -\lambda^1 \cdot C^1 \\ \frac{\Delta C^2}{\Delta t} &= -\lambda^2 \cdot C^2 + Y_{1/2} \cdot \lambda^1 \cdot C^1 \\ &\dots \\ \frac{\Delta C^k}{\Delta t} &= -\lambda^k \cdot C^k + Y_{k-1/k} \cdot \lambda^{k-1} \cdot C^{k-1}\end{aligned}\quad (53)$$

where λ^k specifies the first-order reaction coefficient for species k and $Y_{k-1/k}$ sets the yield coefficient between species k-1 and k. Yield coefficients are based on stoichiometric relations between the species for which relations are specified. The yield coefficients are specified in the Species tab (Figure 117), the reaction coefficients are specified in *Models > MT3D/SEAWAT > Packages > Reaction (RCT) > First-Order Reaction Rate (Dissolved Phase)* and *Models > MT3D/SEAWAT > Packages > Reaction (RCT) > First-Order Reaction Rate (Sorbed Phase)*.

- **Monod Kinetic Reaction:** The concentration change due to the chemical reaction from one transport step to another at a cell can be expressed by

$$\Delta C = -Mic_t \cdot \mu_{max} \cdot \frac{C}{K_s + C} \quad (54)$$

where

Mic_t is the total microbial concentration [ML⁻³] at the cell, K_s is a constant, described by (Alexander, 1999), called the half-saturation constant, and is equal to the constituent concentration [ML⁻³] at which the microbial growth rate is half of μ_{max} . It is important to note that the microbial concentration depends on many chemical and biological considerations and is therefore difficult to estimate. The total microbial concentration is assumed constant in MT3D. The values of $Mic_t \cdot \mu_{max}$ and K_s must be specified by the user and need to be carefully considered during model calibration procedures. The Monod kinetics are available only for dissolved-phase organic compounds and do not consider sorbed concentrations, though reduction in dissolved concentrations resulting from microbial remediation may lead to desorption and a consequent rebound of concentration.

- **Custom Reaction:** See the input instruction of the MT3D-USGS software distribution (Bedekar, et al., 2016) for details about custom reaction modules.
 - **Reaction Module (IREACTION)** defines the custom reaction module. At least two species must be simulated when a custom reaction module is used.
 - **Electron Donor (ED)** is the electron donor in the EA/ED reaction.
 - **Electron Acceptor (EA)** is the electron acceptor in the EA/ED reaction.
 - **Stoichiometric ratio (F):** F is used in the simulated equation as follows: $ED + F \times EA \rightarrow \text{Product}$.
 - **Rec_Filename:** This is the file that stores parameters required for simulating a user-defined kinetic reaction.

5. **VDF Tab** (Figure 119) contains options for the Variable-Density Flow Process of SEAWAT.

- **IWTABLE:** Check the box to activate the variable-density water-table correction. For details, see Equation 82 of (Guo, et al., 2002).
- **MFNADVFD** this defines the method for calculating Internodal density values.
- **NSWTCPL** is used to determine the flow and transport coupling procedure.
 - If NSWTCPL = 0 or 1, flow and transport will be explicitly coupled using a one-timestep lag. The explicit coupling option is normally much faster than the iterative option and is recommended for most applications.
 - If NSWTCPL > 1, NSWTCPL is the maximum number of non-linear coupling iterations for the flow and transport solutions. SEAWAT-2000 will stop execution

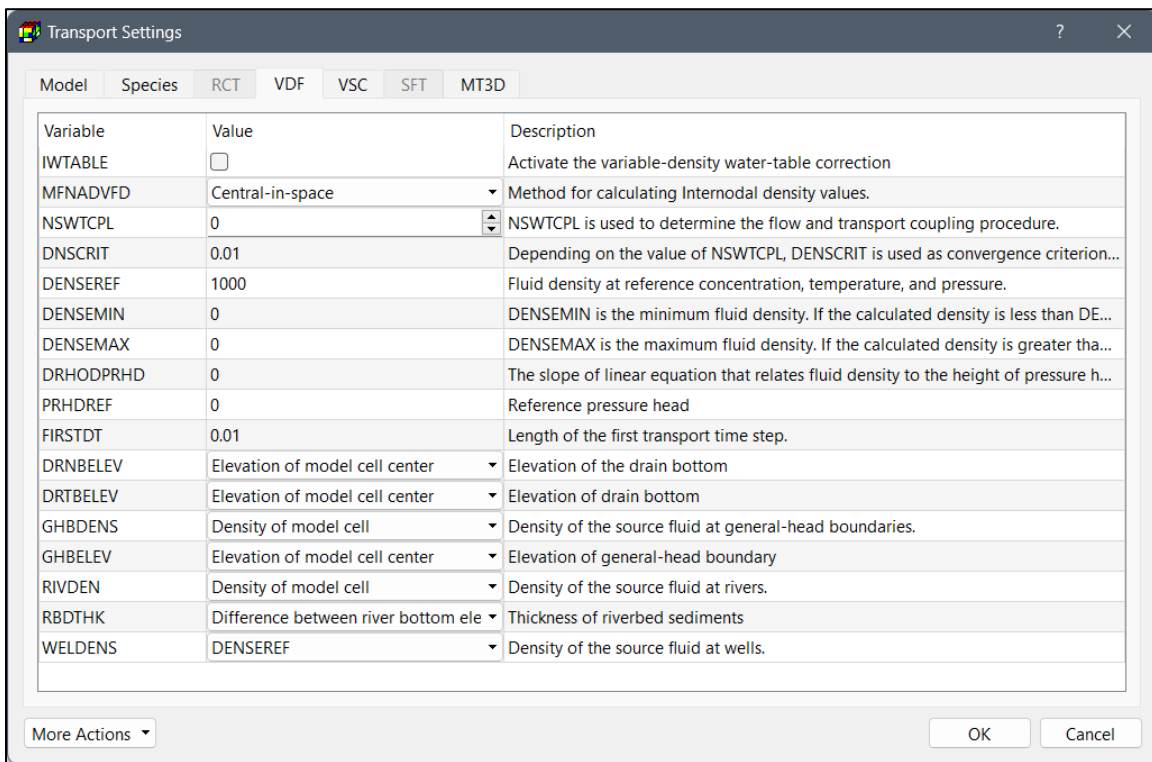


Figure 119. The VDF tab of the Transport Settings dialog box

after NSWTCPL iterations if convergence between flow and transport has not occurred.

- If NSWTCPL = -1, the flow solution will be recalculated only for
 - The first transport step of the simulation, or
 - The last transport step of the MODFLOW time step, or
 - The maximum density change at a cell is greater than DNSCRIT.
- **DNSCRIT:** Depending on the value of NSWTCPL, DNSCRIT is used as convergence criterion or maximum density threshold.
 - If NSWTCPL > 1, DNSCRIT is the convergence criterion for convergence between flow and transport. If the maximum fluid density difference between two consecutive implicit coupling iterations is not less than DNSCRIT, the program will continue to iterate on the flow and transport equations, or will terminate if NSWTCPL is reached.
 - If NSWTCPL is -1, DNSCRIT is the maximum density threshold. If the fluid density change (between the present transport time step and the last flow solution) at one or more cells is greater than DNSCRIT, then SEAWAT will update the flow field (by solving the flow equation with the updated density field).
- **DENSEREF** is the fluid density at reference concentration, temperature, and pressure.
- **DENSEMIN** is the minimum fluid density. If the calculated density is less than DENSEMIN, it is set to DENSEMIN. To remove this limitation, set DENSEMIN to zero.
- **DENSEMAX** is the maximum fluid density. If the calculated density is greater than DENSEMAX, it is set to DENSEMAX. To remove the limitation, set DENSEMAX to zero.
- **DRHODPRHD** is the slope of linear equation that relates fluid density to the height of pressure head.
- **PRHDREF** is the reference pressure head.
- **FIRSTDT** is the length of the first transport time step.
- **DRNBELEV**
 - **User-specified in the DRN Package:** the user-specified values of Elevation of Drain Bottom will be assigned to the auxiliary variable DRNBELEV in the input file to the DRN package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.
 - **Elevation of model cell center:** the auxiliary variable DRNBELEV will not be assigned to the input file to the DRN package, consequently the cell center elevation will be used by the VDF process of SEAWAT.
- **DRTBELEV**
 - **User-specified in the DRT Package:** the user-specified values of Elevation of Drain Bottom will be assigned to the auxiliary variable DRTBELEV in the input file to the DRT package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.

- **Elevation of model cell center:** the auxiliary variable DRTBELEV will not be assigned to the input file to the DRT package, consequently the cell center elevation will be used by the VDF process of SEAWAT.
- **GHBDENS⁶⁰** controls the density of the source fluid at general-head boundaries.
 - **User-specified in the GHB Package:** the user-specified values of Water Density will be assigned to the auxiliary variable GHBDENS in the input file to the GHB package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.
 - **Density of model cell:** the auxiliary variable GHBDENS will not be assigned to the input file to the GHB package and the calculated density of model cells will be used by the VDF process of SEAWAT by default.
- **GHBELEV**
 - **User-specified in the GHB Package:** the user-specified values of GHB Elevation will be assigned to the auxiliary variable GHBELEV in the input file to the GHB package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.
 - **Elevation of model cell center:** the auxiliary variable GHBELEV will not be assigned to the input file to the GHB package and the cell center elevation will be used by the VDF process of SEAWAT by default.
- **RIVDEN⁶⁰** controls the density of the source fluid at rivers.
 - **User-specified in the RIV Package:** the user-specified values of Water Density will be assigned to the auxiliary variable RIVDEN in the input file to the RIV package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.
 - **Density of model cell:** the auxiliary variable RIVDEN will not be assigned to the input file to the RIV package and the calculated density of model cells will be used by the VDF process of SEAWAT by default.
- **RDBTHK**
 - **User-specified in the RIV Package:** the user-specified values of Thickness of Riverbed Sediments will be assigned to the auxiliary variable RDBTHK in the input file to the RIV package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.
 - **Difference between river bottom elevation and cell center:** the auxiliary variable RDBTHK will not be assigned to the input file to the RIV package and the difference between the river bottom elevation and cell center elevation will be used by the VDF process of SEAWAT.
- **WELDENS⁶⁰** controls the density of the source fluid at wells.

⁶⁰ When all Density boxes in the Species tab are cleared (i.e., SEAWAT runs in the uncoupled mode), the user has the option to determine the density of the source fluid. The setting is ignored if the Density box of any of the species is checked.

- **User-specified in the WEL Package:** the user-specified values of Water Density will be assigned to the auxiliary variable WELDENS in the input file to the WEL package and used for the Variable-Density Flow (VDF) process of a SEAWAT simulation.
- **DENSEREF:** the auxiliary variable WELDENS will not be assigned to the input file to the WEL package and the value of DENSEREF will be used by the VDF process of SEAWAT by default.

6. **VSC Tab** (Figure 120) contains Options for the Viscosity Package of SEAWAT.

- **VISCMIN** is the minimum fluid viscosity. If the calculated viscosity is less than VISCMIN, it is set to VISCMIN. To remove this limitation, set VISCMIN to zero.
- **VISCMAX** is the maximum fluid viscosity. If the calculated viscosity is greater than VISCMAX, it is set to VISCMAX. To remove the limitation, set VISCMAX to zero.
- **VISCREF** is the fluid viscosity at the reference concentration and reference temperature. For most simulations, VISCREF is specified as the viscosity of freshwater.
- **MUTEMPOPT** is the option for including the effect of temperature on fluid viscosity.
 - **MUTEMPOPT = 0**, the effect of temperature on fluid viscosity is defined by DMUDC and CMUREF of the species with the Viscosity box checked.
 - **MUTEMPOPT = 1**, fluid viscosity is calculated using equation 18 in (Langevin, et al., 2007). The variable MTMUTEMPSPEC and the coefficients A1 through A4 (see below) must be defined.
 - **MUTEMPOPT = 2**, fluid viscosity is calculated using equation 19 (Langevin, et al., 2007). The variable MTMUTEMPSPEC and the coefficients A1 through A5 (see below) must be defined.

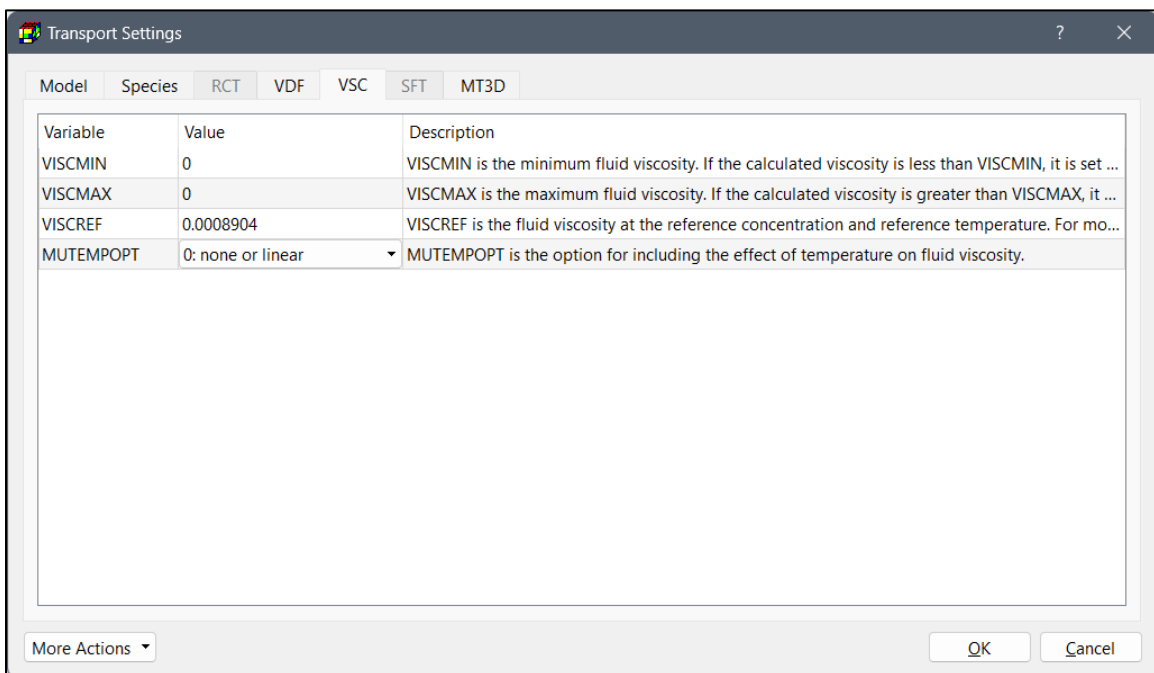


Figure 120. The VSC tab of the Transport Settings dialog box

- **MUTEMPOPT = 3**, fluid viscosity is calculated using equation 20 (Langevin, et al., 2007). The variable MTMUTEMPSPEC and the coefficients A1 and A2 (see below) must be defined.
 - **MTMUTEMPSPEC** is the species that corresponds to temperature. The Viscosity box of this species should NOT be checked.
 - **A1 to A5** are variables used in equations that relate dynamic viscosity to temperature. See MUTEMPOPT above for details.
7. **SFT Tab** (Figure 121) contains options for the Streamflow Transport (SFT) Package of MT3D-USGS.
- **Time Weighting Factor (WIMP)** is the stream solver time weighting factor. WIMP ranges between 0.0 and 1.0. Values of 0.0, 0.5, or 1.0 correspond to explicit, Crank-Nicolson, and fully implicit schemes, respectively.
 - **Space Weighting Factor (WUPS)** is the space weighting factor employed in the stream network solver. WUPS ranges between 0.0 and 1.0. Values of 0.0 and 1.0 correspond to a central-in-space and upstream weighting factors, respectively.
 - **Maximum Number of Iterations (MXITERSF)** is the maximum number of iterations the SFT solver can use to find a solution of the stream transport problem.
 - **Closure Criterion (CCLOSESF)** is the closure criterion for the SFT solver.
 - **Printout Option (IPRTXMD)** controls the printout from the SFT Solver.

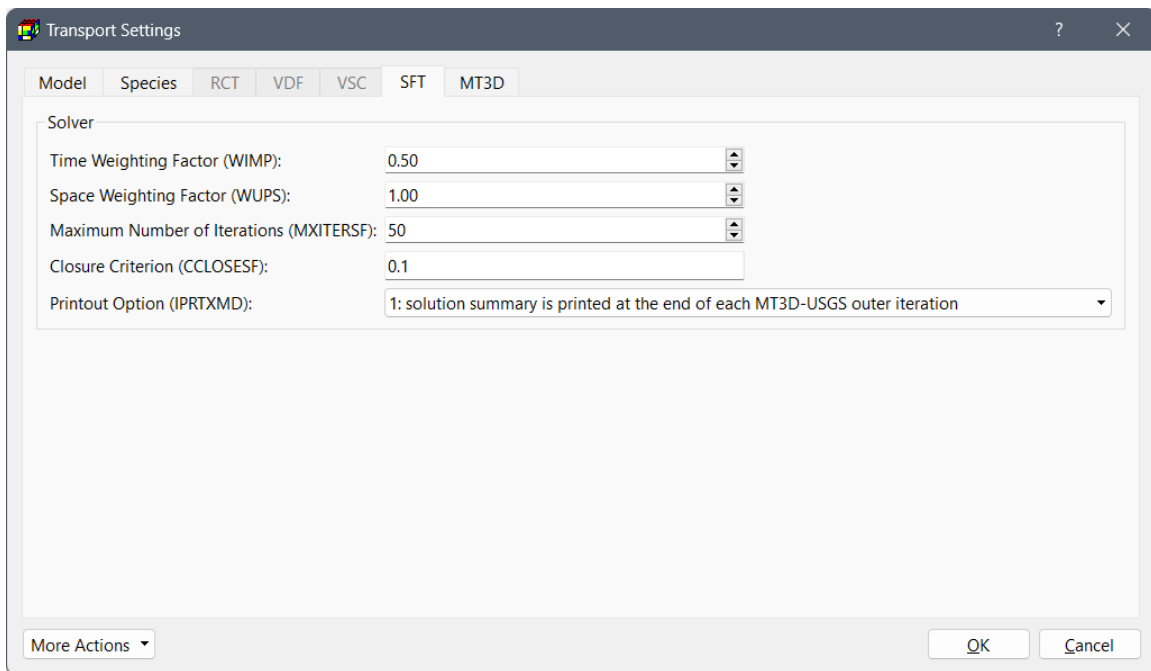


Figure 121. The SFT tab of the Transport Settings dialog box

8. **MT3D Tab** (Figure 122) contains options and control data of MT3D.

- **MODFLOWSTYLEARRAY:** Check this box to write arrays in the MODFLOW-format. Uncheck this box to create MT3DMS-compatible input files.
- **DRYCELL:** The DRYCELL option in MT3D-USGS enables mass transfer through dry cells, allowing solute transport to continue in areas where groundwater flow occurs through previously dewatered cells. The major difference between MT3D-USGS (with the DRYCELL option) and MT3DMS is how they handle solute transport in dry cells:
 - MT3DMS deactivates cells with a saturated thickness below the minimum threshold, resetting their fluxes to zero, effectively stopping solute transport in those areas. This works well with MODFLOW-2005 but not with MODFLOW-NWT, where flow can occur through dry cells.
 - MT3D-USGS (with DRYCELL enabled) allows solute transport through dry cells, ensuring mass transfer continues even when groundwater flow persists in dewatered areas, making it compatible with MODFLOW-NWT.
- **NOWETDRYPRINT:** Check this box to disable the printing of messages indicating the "re-wetting" and "drying" of model cells to the model run listing file.
- **OMITDRYCELLBUDGET:** In MT3D-USGS, the OMITDRYCELLBUDGET option controls whether dry cell budget terms are excluded from the transport simulation output. This option is useful when dealing with large models where dry cell contributions are insignificant, helping to reduce output file size and improve processing efficiency.
- **ALTWTSORB:** In MT3D-USGS, the ALTWTSORB option controls the method used for calculating sorption in the mass storage term when simulating transport with nonlinear or nonequilibrium sorption. This option is particularly useful for improving

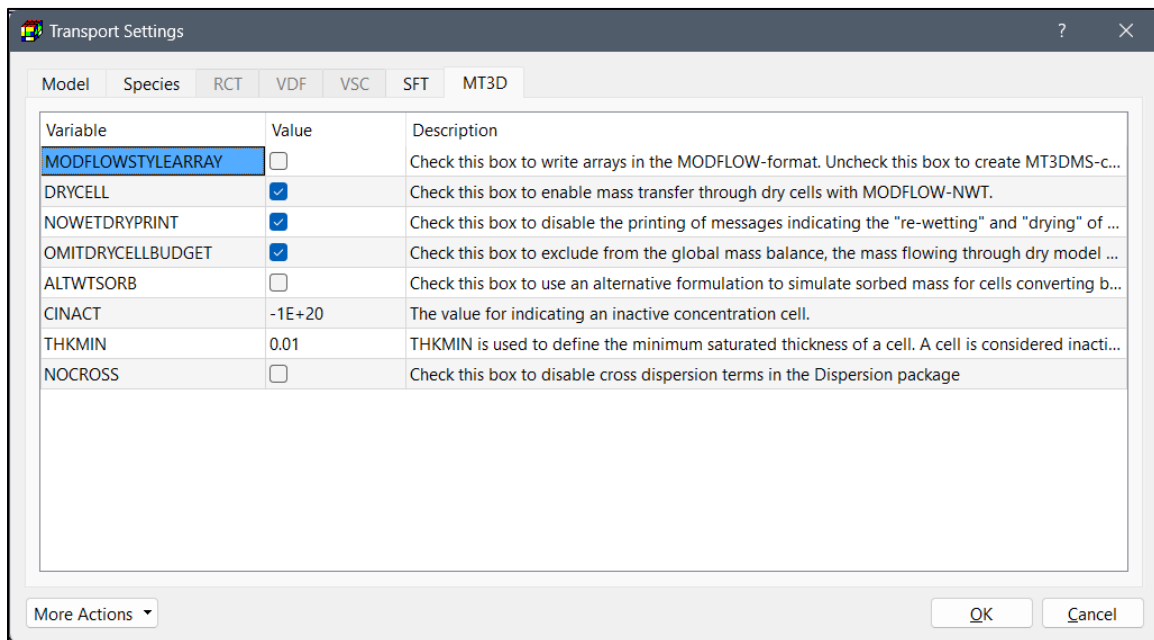


Figure 122. The MT3D tab of the Transport Settings dialog box

accuracy in models with strong sorption effects, ensuring more reliable mass balance calculations in complex transport scenarios.

- If this option is checked: The sorbed concentration is computed using an alternative weighting scheme for better numerical stability and mass conservation in nonlinear or nonequilibrium sorption simulations.
- If this option is not checked: The standard MT3DMS approach is used for updating sorbed concentration.
- **CINACT** is the value for indicating an inactive concentration cell.
- **THKMIN**: In MT3DMS and MT3D-USGS, THKMIN defines the threshold for deactivating a cell in solute transport based on its saturated thickness:
 - If THKMIN > 0, the threshold is a fraction of the model layer thickness (e.g., 0.1 means 10% of the layer thickness).
 - If THKMIN < 0, the absolute value is used as the threshold in model units (e.g., -1.0 means 1 meter or foot, depending on the model's units).

When a cell is deactivated, all advective and dispersive fluxes associated with it are reset to zero, preventing solute movement.

Compatibility with Flow Models

- In MT3DMS, this behavior is suitable for MODFLOW-2005 and earlier versions, where flow ceases in dry cells.
- In MT3D-USGS, if the DRYCELL option is enabled, mass transfer can still occur through dry cells, even if they fall below THKMIN, making it compatible with MODFLOW-NWT.
- **NOCROSS**: Check this box to disable cross dispersion terms in the Dispersion package. For most simulations, you should retain the cross-dispersion terms because they typically improve accuracy. For complex problems resulting in negative concentrations, the NOCROSS option provides a way to determine the cause of the negative concentrations.

3.9.3.2. Initial Concentration

At the beginning of a transport simulation, MT3D and SEAWAT require the initial concentration of each active species at each active concentration cell, i.e. ICBUND > 0.

3.9.3.3. Packages

3.9.3.3.1. Advection (ADV)

The following parameters are defined in the Advection (ADV) Package dialog box (Figure 123).

1. Settings

- **Advection Solution Option (MIXELM):**
 - **Third-order TVD Scheme (ULTIMATE)** is based on the ULTIMATE algorithm (Leonard, 1988), which is in turn derived from the earlier QUICKEST algorithm

(Leonard, 1979). With the ULTIMATE scheme, the solution is mass conservative, without excessive numerical dispersion and artificial oscillation.

- **Finite-Difference Method** is implicit with the Generalized Conjugate Gradient solver (GCG) package. Due to the problems of numerical dispersion and artificial oscillation, the upstream finite difference method is only suitable for solving transport problems not dominated by advection, see p. 22 in (Zheng, et al., 1999) for details. When the Peclet number P_e ($P_e = \frac{\Delta x}{\alpha L}$; Δx is grid spacing and αL is the longitudinal dispersivity) is smaller than two, the upstream finite difference method is reasonably accurate. However, it is advisable to use the upstream finite difference method for obtaining first approximations in the initial stages of a modeling study.
- **Forward Tracking Method of Characteristics (MOC)** has been widely used. One of the most desirable features of the MOC technique is that it is virtually free of numerical dispersion that poses serious difficulty in many numerical schemes. The

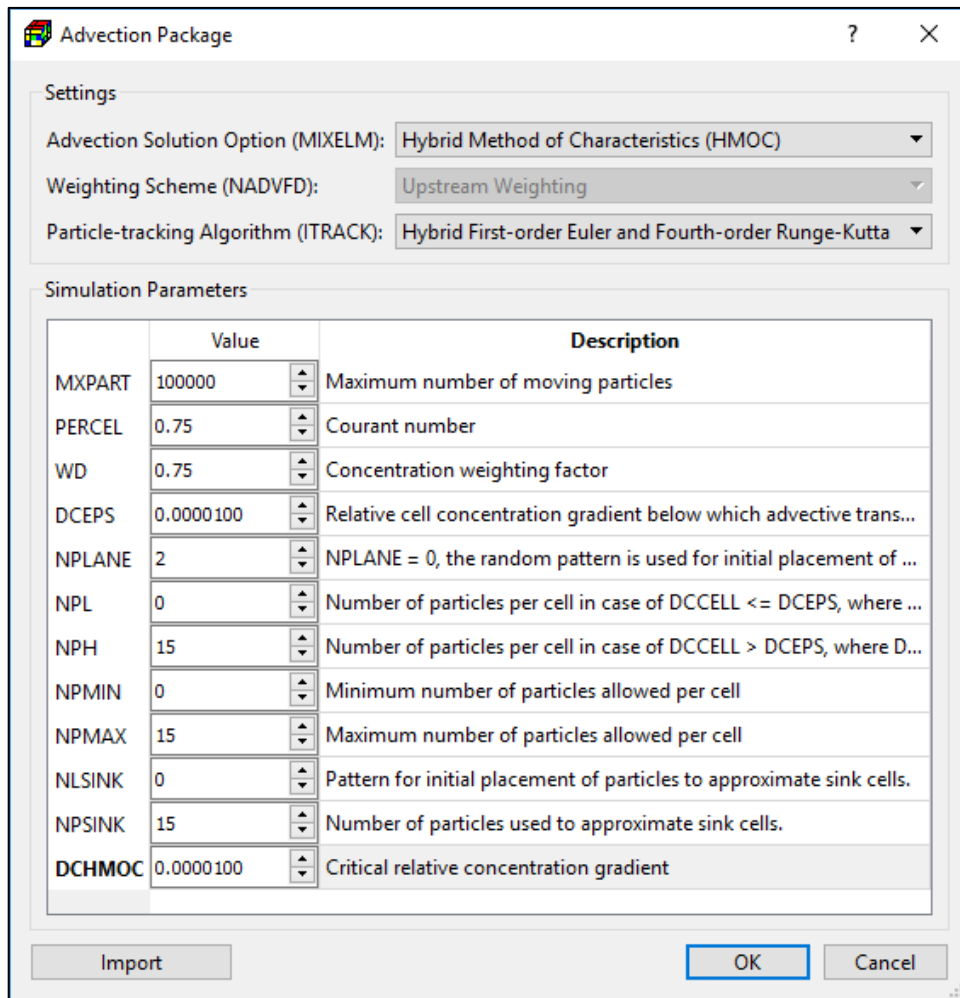


Figure 123. The Advection Package dialog box

major drawback of the MOC scheme is that it can be slow and requires a large amount of computer memory when many particles are required. Also, the computed concentrations sometimes tend to show artificial oscillations.

- **Backward Tracking Method of Characteristics (MMOC)** uses one particle for each finite-difference cell and is normally faster than MOC. At each new time level, a particle is placed at the nodal point of each finite-difference cell. The particle is tracked backward to find its position at the old time. The concentration associated with that position is used to approximate the advection-relevant average concentration at the cell, where the particle is placed. MMOC is free of artificial oscillations if implemented with a lower-order velocity interpolation scheme (such as linear interpolation used MT3D-USGS). However, with a lower-order velocity interpolation scheme, the MMOC technique introduces some numerical dispersion, especially for sharp front problems.
- **Hybrid Method of Characteristics (HMOC)** attempts to combine the strengths of the MOC and MMOC schemes by using an automatic adaptive scheme. The fundamental idea behind the scheme is automatic adaptation of the solution process to the nature of the concentration field. When sharp concentration fronts are present, the advection term is solved by MOC with moving particles dynamically distributed around each front. Away from such fronts, the advection term is solved by MMOC. The criterion for controlling the switch between the MOC and MMOC schemes is given by DCHMOC (see below).
- **Weighting Scheme (NADVFD)** is needed only when the implicit finite-difference method is used. In the finite-difference method, when computing the mass flux into a model cell by advection, the concentration values at the cell interfaces between two neighboring cells are used. For the upstream weighting scheme, the interface concentration in a direction is equal to the concentration at the upstream node along the same direction. For the central-in-space weighting scheme, the interface concentration is obtained by linear interpolation of the concentrations at the two neighboring cells. As pointed by (Zheng, et al., 1999), the central-in-space scheme does not lead to intolerable numerical dispersion when the grid spacing is regular. However, if transport is dominated by advection, the upstream weighting is preferred, as the central-in-space weighting scheme can lead to excessive artificial oscillation.
- **Particle-tracking Algorithm (ITRACK)** is used in combination with the method of characteristics.
 - **First-order Euler:** Using this algorithm, numerical errors tend to be large unless small transport steps are used. The minimum allowed transport step of all particles is determined by MT3D-USGS and is used in a transport step.
 - **Fourth-order Runge-Kutta:** The basic idea of this method is to calculate the particle velocity four times for each tracking step: one at the initial point, twice at two trial midpoints, and once at a trial end point. A weighted velocity based on values evaluated at these four points is used to move the particle to a new position. This method permits the use of larger tracking steps. However, its computational effort is considerably larger than the first-order Euler method.

- **Hybrid First-order Euler and Fourth-order Runge-Kutta:** The option is implemented by automatic selection of the fourth-order Runge-Kutta algorithm for particles located in cells which contain or are adjacent to sinks or sources, and automatic selection of the first-order Euler algorithm for particles located elsewhere.
2. **Simulation Parameters:** Depends on the selected Solution Scheme, one or more of the following parameters may be required.

- **MXPART:** Maximum number of moving particles in a simulation.
- **PERCEL** is the number of cells (or a fraction of a cell) any particle will be allowed to move in any direction in one transport step. Generally, $0.5 \leq \text{PERCEL} \leq 1$.
- **WD** is Concentration-weighting factor. WD lies between 0 and 1. The value of 0.5 is normally a good choice. This number can be adjusted to achieve better mass balance. Generally, it can be increased toward 1 as advection becomes more dominant.
- **DCEPS** is the relative cell concentration gradient below which advective transport is considered negligible. A value around 10^{-5} is generally adequate.
- **NPLANE:**
 - If NPLANE = 0, the random pattern is used for initial placement of particles.
 - If NPLANE > 0, the fixed pattern is selected for initial placement. The value of NPLANE serves as the number of “planes” on which initial particles are placed. For two-dimensional simulations in plan view, set NPLANE = 1. For cross sectional or three-dimensional simulations, NPLANE = 2 is normally adequate. Increase NPLANE if more resolution in the vertical direction is desired.
- **NPL** is the number of initial particles per cell to be placed at cells where the relative cell concentration gradient $DCCELL_{k,i,j}$ (see equation below) is less than or equal to DCEPS.

$$DCCELL_{k,i,j} = \frac{CMAX_{k,i,j} - CMIN_{k,i,j}}{CMAX - CMIN} \quad (55)$$

where

$CMAX_{k,i,j}$ and $CMIN_{k,i,j}$ are the maximum and minimum concentration values in the immediate vicinity of the cell [k, i, j]; $CMAX$ and $CMIN$ are the maximum and minimum concentration values in the entire grid, respectively.

Generally, NPL can be set to zero since advection is considered insignificant when the $DCCELL_{k,i,j}$ is less than or equal to DCEPS. Setting NPL equal to NPH causes a uniform number of particles to be placed in every cell over the entire grid.

- **NPH** is the number of initial particles per cell to be placed at cells where $DCCELL_{k,i,j}$ is greater than DCEPS. The selection of NPH depends on the nature of the flow field and the computer memory limitation. Generally, a smaller number should be used in relatively uniform flow fields and a larger number should be used in relatively nonuniform flow fields. values exceeding 16 in two-dimensional simulation or 32 in three-dimensional simulation are rarely necessary.

- **NPMIN** is the minimum number of particles allowed per cell. If the number of particles in a cell at the end of a transport step is fewer than NPMIN, new particles are inserted into that cell to maintain enough particles. NPMIN can be set to zero in relatively uniform flow fields and to a number greater than zero in diverging/converging flow fields. Generally, a value between zero and four is adequate.
- **NPMAX** is the maximum number of particles allowed per cell. If the number of particles in a cell exceeds NPMAX, all particles are removed from that cell and replaced by a new set of particles equal to NPH to maintain mass balance. Generally, NPMAX can be set to approximately two times of NPH.
- **NLSINK** is a flag indicating whether the random or fixed pattern is selected for initial placement of particles to approximate sink cells in the MMOC scheme. The convention is the same as that for NPLANE. It is generally adequate to set NLSINK equivalent to NPLANE.
- **NPMAX** is the maximum number of particles allowed per cell. If the number of particles in a cell exceeds NPMAX, all particles are removed from that cell and replaced by a new set of particles equal to NPH to maintain mass balance. Generally, NPMAX can be set to approximately two times of NPH.
- **DCHMOC** is the critical relative concentration gradient for controlling the selective use of either MOC or MMOC in the HMOC solution scheme. The MOC solution is selected at cells where $DCCELL_{k,i,j} > DCHMOC$. The MMOC solution is selected at cells where $DCCELL_{k,i,j} \leq DCHMOC$.

3.9.3.3.2. Dispersion (DSP)

1. **Longitudinal Dispersivity:** Select this item to define the longitudinal dispersivity of the model cells with the Data Editor. Longitudinal dispersivity is used to approximate the spreading of the solute concentration in groundwater caused by the irregular shape of the interconnected pore space and the velocity variations at the microscopic level as well as the unresolved macroscopic level. The velocity of groundwater varies according to the size of the pores and water moves faster at the internal points between soil grains than on the solid surface. This spreading is often referred to as mechanical dispersion and it occurs in all three spatial directions. Values of dispersivity used for simulations generally depend on the scale of a concentration plume being considered. While a plume “grows”, it will not only undergo the microscopic mechanical dispersion but also the dispersion caused by macroscopic heterogeneities. This results in a trend of increasing dispersivity values with the scale of observation. Summaries of the scale-dependent dispersivity values can be found in (Anderson, 1984), (Gelhar, et al., 1992), and (Spitz, et al., 1996). Note that all heterogeneity, which is not explicitly represented in the model, should be incorporated into the dispersion coefficients.
2. **Diffusion Coefficient:** Select this item to define the effective molecular diffusion coefficient of the model cells with the Data Editor. describes the diffusive flux of a solute in water from an area of greater concentration toward an area where it is less concentrated. The molecular diffusion coefficient is generally very small and negligible

compared to the mechanical dispersion and is only important when groundwater velocity is very low.

3.9.3.3.3. Reaction (RCT)

This menu is used for specifying the required parameters associated with the sorption and/or kinetic rate reaction types selected in Transport Settings dialog box (Figure 118). Below is a complete list of all items of this menu. The available items vary with the selected sorption and kinetic rate reaction types.

- 1. Initial Concentration (Sorbed Phase):** Defines the initial Concentration for the sorbed phase.
- 2. Initial Concentration (Immobile Liquid Phase):** Defines the initial Concentration for the immobile liquid phase.
- 3. Distribution Coefficient:** Defines the distribution coefficient
- 4. Distribution Coefficient (Mobile Domain):** Defines the distribution coefficient for the mobile domain.
- 5. Distribution Coefficient (Immobile Domain):** Defines the distribution coefficient for the immobile domain
- 6. Freundlich equilibrium constant:** Defines the Freundlich equilibrium constant.
- 7. Freundlich equilibrium constant:** Defines the Freundlich exponent A.
- 8. Langmuir equilibrium constant:** Defines the Langmuir equilibrium constant.
- 9. Langmuir equilibrium constant:** Defines the Sorption Capacity for the Langmuir sorption.
- 10. Porosity (Immobile Domain):** Defines the porosity of the immobile domain.
- 11. Mass Transfer Rate (Between Phases):** Defines the first-order mass transfer rate between dissolved and sorbed phases.
- 12. Mass Transfer Rate (Between Domains):** Defines the first-order mass transfer rate between mobile and immobile domains.
- 13. First-Order Reaction Rate (Dissolved Phase):** Defines the first-order reaction rate for the dissolved phase.
- 14. First-Order Reaction Rate (Sorbed Phase):** Defines the first-order reaction rate for the sorbed phase.
- 15. Zero-Order Reaction Rate (Dissolved Phase):** Defines the zero-order reaction rate for the dissolved phase.
- 16. Zero-Order Reaction Rate (Sorbed Phase):** Defines the zero-order reaction rate for the sorbed phase.
- 17. Bacterium Growth Rate (Monod kinetics):** Defines the product of the total microbial concentration and maximum specific growth rate of the bacterium.
- 18. Half-Saturation constant Ks (Monod kinetics):** Defines the half-saturation constant Ks for Monod kinetics.

3.9.3.3.4. Sink/Source Mixing (SSM)

This menu is used for specifying the concentration associated with the fluid of spatially distributed sources or sinks. Menu items of this menu are deactivated if the corresponding hydraulic features of *Models > MODFLOW > Packages* menu are not used (checked). The specified concentration will be used in the transport simulation if the corresponding menu item is checked. If a checked item is no longer necessary for the transport simulation, simply select the item again and deactivate it. If the concentration of a source or sink is not specified, the default value of zero is used.

Concentration values of point sources, including DRT1, GHB, FHB, MNW1, MNW2, RES1, RIV, STR, and WEL, are specified using the CBC, Polyline, and Polygon input methods of the respective packages.

Evapotranspiration and Unsaturated Zone Seepage Flux are the only sinks whose concentration can be specified. The concentration of a sink cannot be greater than that of the groundwater at the sink cell. If the sink concentration is specified greater than that of the groundwater, it is automatically set equal to the concentration of the groundwater. Therefore, setting a large sink concentration value (e.g., 1×10^{30}) to evapotranspiration ensures that the groundwater concentration is used for the computation.

The menu item Time-Variant Constant Concentration allows you to define cells with specified concentration values that vary over time. A time-variant constant concentration cell is defined by assigning a non-zero concentration value in the Data Editor. In a multiple stress period simulation, once constant-concentration condition is assigned to a cell, it remains so during the simulation, but its concentration value may vary in different stress periods. In a multispecies simulation, the constant-concentration condition applies to all species. If the constant-concentration condition does not apply to a species, assign a negative concentration value for that species.

3.9.3.3.5. Variable-Density Flow (VDF) > Prescribed Fluid Density

Use this menu item to define the fluid density for SEAWAT. This option is available only when the density effect of all species is turned off. See Species tab (Figure 117 on page 212) for details.

3.9.3.3.6. Viscosity (VSC) > Prescribed Viscosity

Use this menu item to define the fluid viscosity for SEAWAT. This option is available only when the effects of viscosity variations of all species is turned off. See Species tab (Figure 117 on page 212) for details.

3.9.3.4. GCG Solver

The GCG Solver is a general-purpose iterative solver based on the generalized conjugate gradient method for solving the system of the transport equations. A detailed description of the method can be found in (Zheng, et al., 1999). This solver must always be activated. Using this solver,

dispersion, sink/source, and reaction terms are solved implicitly without any stability constraints on the transport step size. The required settings and parameters for this package are specified in the Generalized Conjugate Gradient (GCG) dialog box (Figure 124).

1. Preconditioning Method

(ISOLVE): The GCG package has three preconditioning options, Jacobi, SSOR, and Modified Incomplete Cholesky. The last one usually takes less iterations than the other methods, but it requires significantly more memory.

2. Iteration Parameters:

- **Maximum Number of Outer Iterations (MXITER)** and **Maximum Number of Inner Iterations (ITER1):** The GCG solver

has two iteration loops, an inner and an outer loop. Similar to the PCG solver of MODFLOW, all coefficients in the (transport-) matrix \underline{A} and the right-hand-side vector \underline{b} remain unchanged during inner iterations. The inner loop continues until ITER1 – iterations are executed, or the convergence criterion is met. If some of the coefficients in \underline{A} are dependent on the concentration being solved, as in the case of nonlinear sorption, they must be updated in outer iterations. Therefore, MXITER should be set greater than one only when a nonlinear sorption isotherm is included in the simulation. For ITER1, a value between 30 and 50 should be adequate for most problems.

- **Relaxation Factor (ACCL)** is only used for the SSOR option; a value of 1.0 is generally adequate.
- **Dispersion Tensor Cross Terms Option (NCRS):** NCRS is a flag for treatment of dispersion tensor cross terms. NCRS=0, lump all dispersion cross terms to the right-hand side of the system of transport equations. It is approximate but highly efficient. NCRS=1, include full dispersion tensor. This option is memory intensive. However, for critical applications the full dispersion tensor should be included.

- 3. Concentration Closure Criterion (CCLOSE)** is the convergence criterion in terms of relative concentration. A value between 10^{-6} and 10^{-4} is generally adequate. Before solving the system of transport equations, it is normalized by dividing the concentration terms by the maximum concentration of all cells. When the change of the normalized concentration at all cells during an inner iteration is less than or equal to CCLOSE, iteration stops. When it

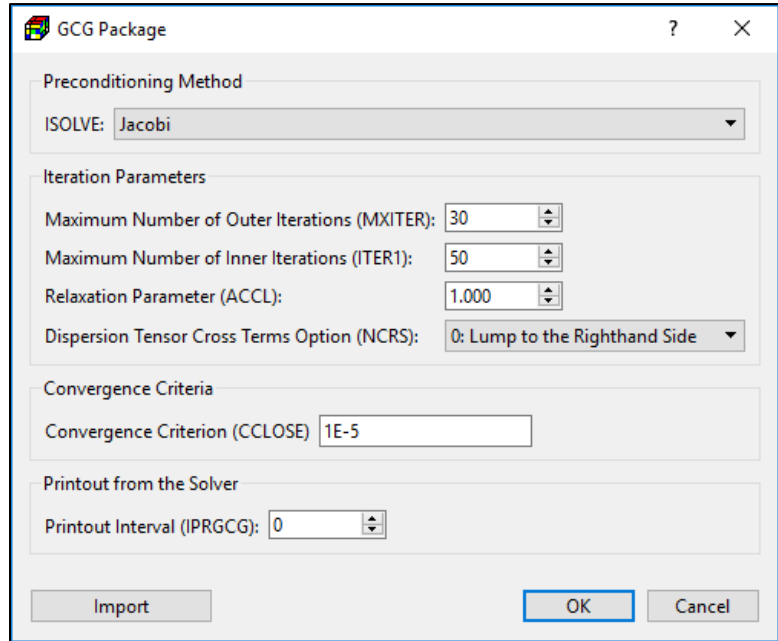


Figure 124. The GCG Package dialog box

takes only one inner iteration to converge, the solution is considered to have converged and the simulation proceeds to the next transport step.

4. **Printout Interval (IPRGCG):** The maximum concentration changes are printed out whenever the iteration number is an even multiple of this printout interval. Set it to zero for printing only at the end of each stress period.

3.9.3.5. Output Control

The available options of the Output Control (MT3D/SEAWAT) dialog box are described as follows.

1. Output Options Tab (Figure 125)

- **SAVUCN** indicates whether the calculated concentration values should be saved in binary files which are used by PM for visualization. The default file names of the binary files are MT3Dnnn.UCN for the dissolved phase and MT3Dnnns.UCN for the sorbed phases, where nnn is the species number.
- **IFMTCN** indicates whether the calculated concentration values should be printed to the Run Listing file.
- **IFMTNP** indicates whether the number of particles in each cell should be printed to the Run Listing file.
- **IFMTRF** indicates whether the model-calculated retardation factor should be printed to the Run Listing file.
- **IFMTDP** indicates whether the model-calculated, distance weighted dispersion coefficient should be printed to the Run Listing File.
- **CHKMAS** indicates whether mass budget summary information should be printed to the mass budget file. The default file names of the mass budget summary files are MT3Dnnn.MAS, where nnn is the species number.
- **NPRMAS:** Mass budget summary information is printed every NPRMAS transport step, if CHKMAS is checked.
- **NPRS:** If NPRS < 0, simulation results are saved every NPRS transport steps (note that transport step counter starts at 0 within each flow time step.) If NPRS = 0, simulation results are saved at the end of each stress period. If NPRS > 0, simulation results are saved at the time points defined in Output Times.

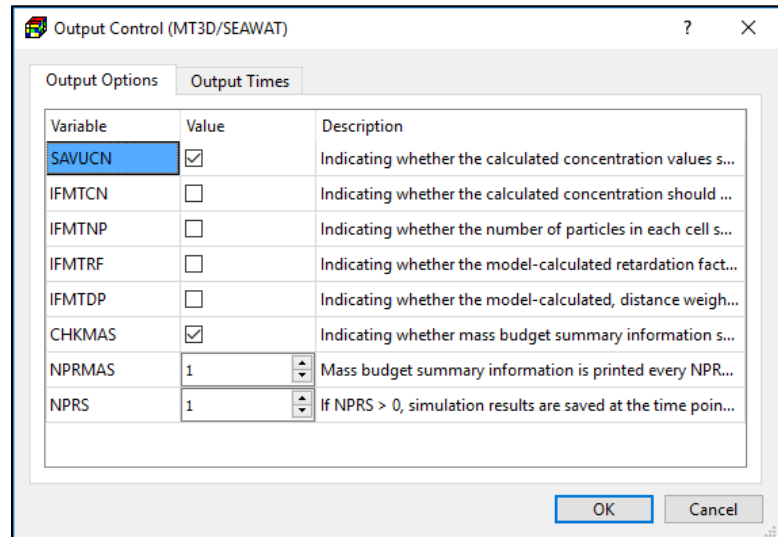


Figure 125. The Output Options tab of the Output Control (MT3D/SEAWAT) dialog box

2. Output Times Tab (Figure 126)

- **Table:** The table contains the times at which the simulation results are saved and/or printed. Time values are expressed in the simulation time and can be edited.

- **Start Time, End Time, and Time Interval:** You can generate a list of output times by specifying the Start Time, End Time, and Time Interval, then clicking the Apply button. Once the list of

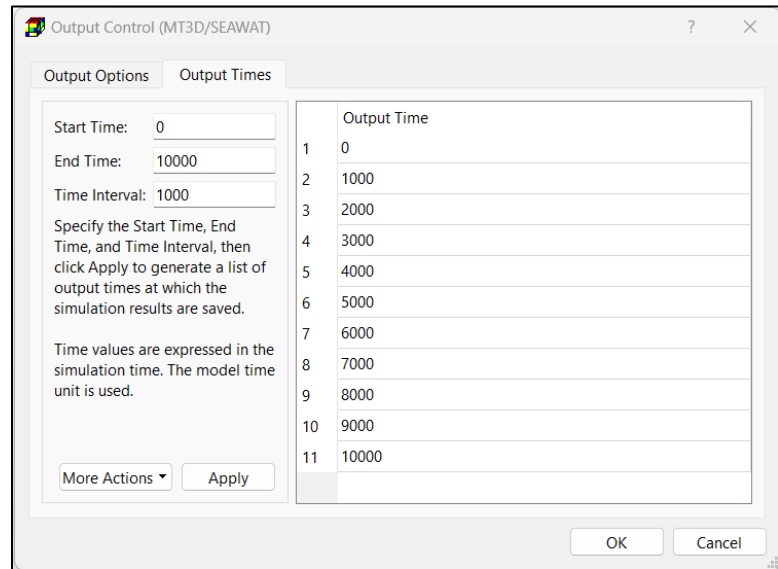


Figure 126. The Output Times tab of the Output Control (MT3D/SEAWAT) dialog box

- **More Actions:**
 - **Set Default:** Select this option to set Start Time to zero, End Time to the total simulation length of the flow simulation, Time Interval to one tenth of the total simulation length, and then populate the table accordingly.
 - **Every Stress Period:** Select this option to assign the end of every stress period to the table in the Output Times tab.
 - **Every Time Step:** Select this option to assign the end of every time step of every stress period to the table in the Output Times tab.

3.9.3.6. Inspect Model Data

Select this menu item to check the correctness and consistency of the model data. The inspection report will be saved in the *mt3d_inspection_log.txt* or *seawat_inspection_log.txt* file.

3.9.3.7. Run

Select this menu item to open the Run MT3D dialog box (Figure 127) or Run SEAWAT dialog box according to the transport model selected in the Transport Settings dialog box (Section 3.9.3.1). The Run MT3D or Run SEAWAT dialog box can be used to start a transport simulation or to build input files. The available settings of the dialog box are described below.

1. Table

- **The build box** . Prior to running a transport simulation, PM uses the user-specified data to build input files for MT3D (or SEAWAT). An input file will be built if its build

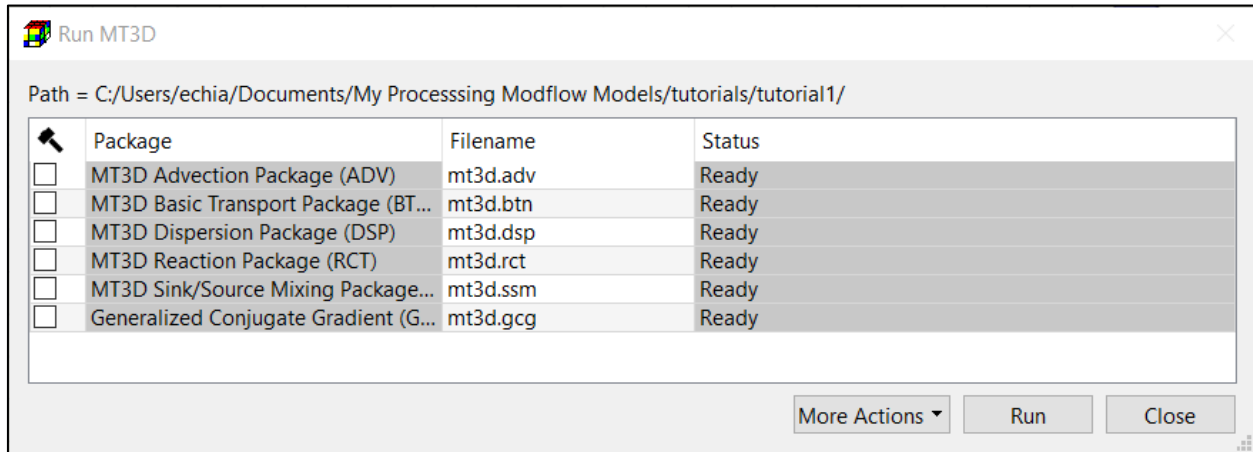


Figure 127. The Run MT3D dialog box

box is checked. If an input file does not exist or the input data has been modified since the last build, its build box will be checked automatically.

- **Package** is the name of the package of the flow model.
- **Filename** is the name of the input file.
- **Status** shows the status (such as Ready, Not Exist, etc.) of the input file.

2. More Actions

- **Inspect Model Data:** Select this action to check the correctness and consistency of the model data. The inspection report will be saved in the *mt3d_inspection_log.txt* or *seawat_inspection_log.txt* file.
- **Build Input Files:** Select this action to build the input files with the checked build boxes. The name file *mt3d.nam* or *seawat.nam* will also be built. The build log is stored in the *mt3d_build_log.txt* or *seawat_build_log.txt* file.
- **Rebuild All Input Files:** Select this action to rebuild all input files, except for existing input files that are marked as read-only. The build log is stored in the *mt3d_build_log.txt* or *seawat_build_log.txt* file.

3. Run: Click this button to build input files with the checked build boxes and the start the transport simulation.

- If *Constant Density Transport (MT3D-USGS)* is selected as the transport model, the simulation will be carried out by the executable file of MT3D-USGS defined in the Preferences dialog box (Figure 37). The transport process will be density-independent.
- If *Constant Density Transport (MT3DMS)* is selected as the transport model, the simulation will be carried out by the executable file of MT3DMS defined in the Preferences dialog box (Figure 37). The transport process will be density-independent.
- If *Variable Density Flow and Transport (SEAWAT)* is selected as the transport model, the simulation will be carried out the executable file of SEAWAT defined in the Preferences dialog box (Figure 37). The transport process can be density-dependent.

The transport simulation progress will be displayed in the MT3D-USGS, MT3DMS or SEAWAT Output window. You may terminate the simulation at any time by clicking on the Terminate button.

3.9.3.8. View

3.9.3.8.1. Run Listing File

Select this menu item to use the Text Editor (see Section 3.5.12) to display the run listing file mt3d.list or seawat.list.

3.9.3.8.2. Scatter Chart

Select *View > Scatter Chart* to display the scatter chart based on the concentration data of the dissolved phase. The use of the Concentration Scatter Chart dialog box is identical as the Head Scatter Chart dialog box (see Section 3.9.1.6.3), with the following exceptions:

1. Species dropdown box is displayed on the lower-left corner of the dialog box. Use this dropdown box to select a species of interest.
2. The simulated concentration data of the selected species is used in the Chart, the Data@SimTime, and Data@ObsTime tabs.

3.9.3.8.3. Line Chart

Select *View > Line Chart* to display the time-series chart based on the concentration data of the dissolved phase. The use of the Concentration Line Chart dialog box is identical as the Head Line Chart dialog box (see Section 3.9.1.6.4), with the following exceptions:

1. Species dropdown box is displayed on the lower-left corner of the dialog box. Use this dropdown box to select a species of interest.
2. The simulated concentration data of the selected species is used in the Chart, the Data@SimTime, and Data@ObsTime tabs.

3.9.3.8.4. Mass Budget Summary

Select this menu item, and then select a species to display the mass budget summary file of that species. The default file names of the mass budget summary files are MT3Dnnn.MAS, where nnn is the species number. Refer to (Neville, 2006) for details about interpreting the MT3D mass budget.

3.9.3.8.5. Concentration Summary

Select this menu item, and then select a species to create a time-history of the average concentration of the model. The calculation is based on the simulated concentration values (stored in the MT3Dnnn.ucn file, where nnn is the number of a species) and the water volume (see Section 3.9.1.6.5 for details) of the model. The calculation results are stored in a comma-separated value file consisting of a number of records. Each record includes the following terms.

- **SimulationTime** is the simulation time at which the simulated concentration values are stored in the MT3Dnnn.ucn file. Since the total amount of water in the model must be calculated as well, the average concentration will not be calculated for the SimulationTime that does not lie between the date/time of the first and last head results.
- **Period** is the stress period to which SimulationTime belongs.
- **TimeStep** is the time step within the given Period to which SimulationTime belongs.
- **Date** is the calendar date in the form of year-month-day. The date corresponds to the given simulation time.
- **Volume** [L³] is the total amount of water in the model. This value is calculated in the same way as the **Volume** described in Section 3.9.1.6.5.
- **Mass** [L³ x concentration unit] is the calculated as follows.

$$Mass = \sum_{i=0}^n (vs_i + vsy_i) \times c_i$$

where

n is the total number of model cells,

c_i is the solute concentration of the selected species in the i^{th} cell at the given simulation time,

vs_i is the volume of water stored in the i^{th} cell due to compressibility, and

vsy_i is the volume of water stored in the pore space of the i^{th} cell.

The values of vs_i and vsy_i are calculated in the same way as described in Section 3.9.1.6.5. It is assumed that vs_i and vsy_i have the same concentration c_i .

- **Concentration** is calculated as Mass/Volume and has the same unit as your input concentration values.

3.9.3.8.6. Production Concentration Summary

Select this menu item, and then select a species to create a time-history of the concentration of the extracted groundwater at the pumping wells defined by the WELL package. The calculation is based on the simulated concentration values (stored in the MT3Dnnn.ucn file, where nnn is the number of a species) and the pumping rates at the well cells specified in the WELL package. The calculation results are stored in a comma-separated value file consisting of a number of records. Each record includes the following terms.

- **SimulationTime** is the simulation time since the start of the simulation at which the simulated concentration values are stored in the MT3Dnnn.ucn file. Since the total amount of water in the model must be calculated as well, the average concentration will not be calculated for the SimulationTime that does not lie between the date/time of the first and last flow results.
- **Period** is the stress period where SimulationTime belongs to.
- **TimeStep** is the time step within the given Period where SimulationTime belongs to.
- **Date** is the calendar date in the form of year-month-day. The date corresponds to the given simulation time.

- **Well** is the well name. The cells with the same well name will be considered a part of the well. If the well name of a well cell is not specified, the default name “row_column” will be used, where row/column are the model row and column of the well cell. Consequently, the cells in various layers of that row/column location will be considered a part of that well.
- **PumpingRate** [L^3/T] is the sum of the pumping rates of the model cells that belong to the given well.
- **MassRate** [$L^3/T \times \text{concentration unit}$] is the sum of [pumping rate \times concentration] of the model cells that belong to the given well.
- **Concentration** is the concentration of the water extracted by the well. The value is calculated as [Concentration = MassRate / PumpingRate] and has the same unit as your input concentration values.

3.9.4. GSFLOW

GSFLOW (Markstrom, et al., 2008; Regan, et al., 2020) is a coupled surface water and groundwater flow model based on the integration of PRMS (Markstrom, et al., 2015) and MODFLOW-NWT, respectively. GSFLOW was developed to simulate coupled surface-water/groundwater flow by simultaneously simulating flow across the land surface, within streams and lakes, and within subsurface saturated and unsaturated materials. Climate data consisting of measured or estimated precipitation, air temperature, and solar radiation, as well as groundwater stresses (such as withdrawals) and boundary conditions (such as prescribed flow) are the hydrologic stresses for a GSFLOW simulation.

Since GSFLOW and PRMS use numerous input and output variables, whenever possible PM uses the variable names given in the GSFLOW manual (Markstrom, et al., 2008), Appendix 1 of the PRMS-IV user guide (Markstrom, et al., 2015), and the GSFLOW input instructions. These documents can be downloaded along with the GSFLOW/RPMS/MODFLOW codes from the USGS website.⁶¹

3.9.4.1. MODFLOW Packages and GSFLOW

To couple MODFLOW with PRMS, several packages of the coupled MODFLOW-NWT code were modified or excluded from GSFLOW. Most noticeable modification is the Unsaturated-Zone Flow (UZF1) Package (Niswonger, et al., 2006a) that was initially developed for GSFLOW to simulate groundwater recharge and discharge from/to land surface by accounting for vertical flow through an unsaturated zone. The second noticeable modification is the Streamflow-Routing (SFR2) Package, which is modified to receive surface runoff and interflow from hydrologic response units (see next section for details). The Lake (LAK3) Package is also modified to interact with PRMS. For details of the MODFLOW modifications, refer to the GSFLOW manual (Markstrom, et al., 2008). According to the GSFLOW manual, the following MODFLOW packages do not have the ability to interact with PRMS, and should not be used in a GSFLOW simulation:

1. Recharge (RCH)⁶², Evapotranspiration (EVT), and Evapotranspiration Segments (ETS1) packages—Use the UZF1 Package instead.
2. The River (RIV)⁶², Stream (STR), Drain (DRN)⁶², and Drain and Return Flow (DRT1) packages—Use the SFR2 Package instead.
3. Reservoir (RES1) Package—Use the LAK3 Package instead.
4. Interbed Storage (IBS1) and Subsidence and Aquifer-System Compaction (SUB) packages—No appropriate replacements available at the time of this writing.

⁶¹ <https://www.usgs.gov/software/coupled-ground-water-and-surface-water-flow-model-gsflow>

⁶² The GSFLOW version distributed with PM was modified to allow for using the RCH package to simulate specified flow boundary conditions, the DRN package to simulate drainage, and the RIV package to simulate river leakage while ignoring the PRMS interaction.

3.9.4.4. Climate Forcing Datasets

GSFLOW requires daily time-series of climate forcing datasets throughout the entire simulation period. Time-series of daily precipitation and minimum and maximum air temperature are required for a GSFLOW simulation; additional data are optional based on the selected module of the PRMS processes (Section 3.9.4.11). For example, wind speed and humidity time-series are required by the **potet_pm_sta** module of the Potential Evapotranspiration process to compute the potential evapotranspiration by using the Penman-Monteith formulation (Penman, 1948; Monteith, 1965).

The observation stations and their daily time-series are entered with the Observation Tool (Section 3.6.3). All daily time-series data must begin at the Start Date of the flow model defined in Figure 2. Observation stations are then associated with HRUs by using the HRU Data dialog box. For example, Figure 129 allows you to associate wind speed stations (**hru_windspeed_sta**) with HRUs.

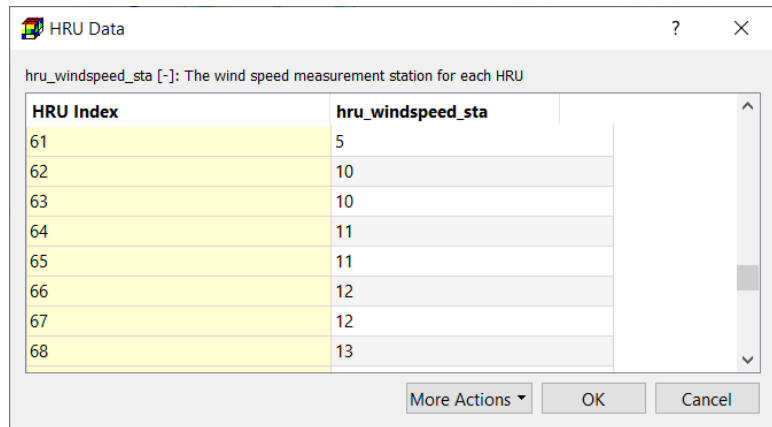


Figure 129. The HRU Data dialog box for wind speed stations

Since the observation tool does not check for missing daily data, it is recommended to use Excel to prepare the input datasets and then import the prepared datasets to the Observation Tool.

3.9.4.5. Settings

The first step to use GSFLOW is to define the simulation options with the GSFLOW Settings dialog box (Figure 130). The available simulation options are detailed below.

- **model_mode:** This is a flag to indicate the simulation mode of GSFLOW. Although GSFLOW supports other modes, such as MODFLOW for MODFLOW-Only simulations, PM only supports the GSFLOW mode for integrated surface water and groundwater simulation.
- **start_time:** This is the GSFLOW simulation start date. To avoid compatibility and inconsistency problems with other software packages, this date should be set the same as the MODFLOW simulation start date, although GSFLOW allows a different start date that is equal or later than the MODFLOW start date.

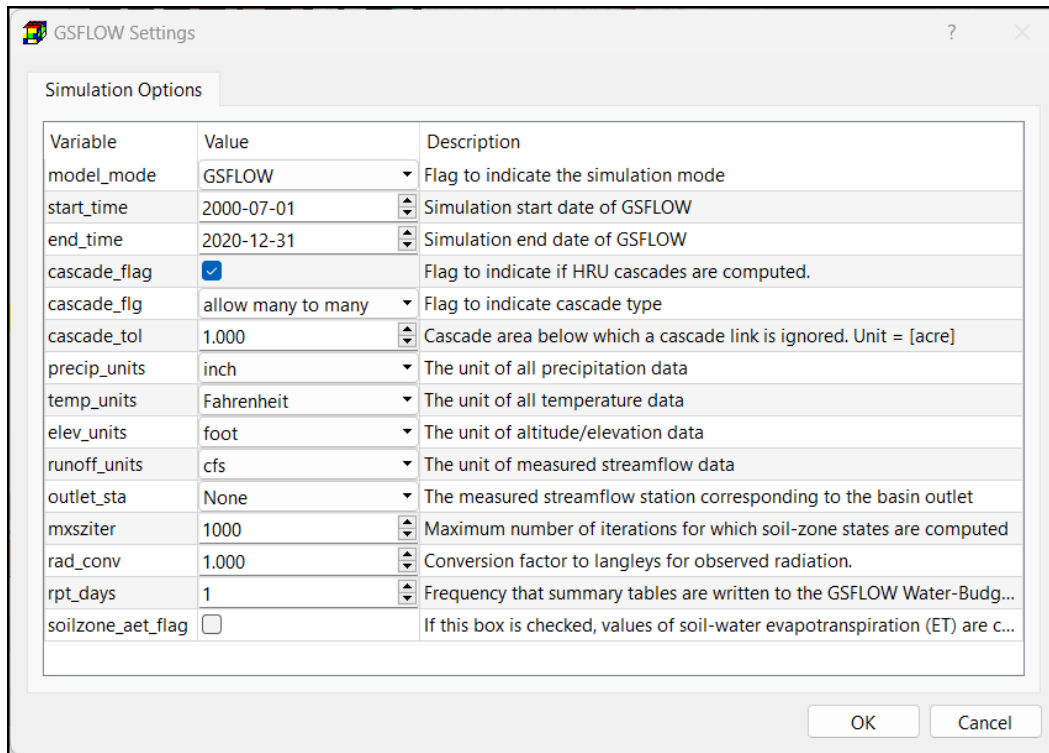


Figure 130. The GSFLOW Settings dialog box

- **end_time:** This is the GSFLOW simulation end date. It must be equal to or earlier than the MODFLOW simulation end date. Start_time and end_time together define the GSFLOW simulation period. Both start_time and end_time must be between the earliest beginning and latest ending date of the time-series data (such as tmax, tmin, precip, and other optional data) used in the simulation.
- **cascade_flag:** This is a flag to indicate if HRU cascading flows are computed. If this box is checked, the cascade paths defined in Models > GSFLOW > Cascade Flow Links will be used for calculating cascading flow.
- **cascade_flg:** This is a routing switch. The option “allow many to many” permits multiple routing paths of inflows and outflows among HRUs (many-to-many routing) and to stream segments. The option “force one to one” forces routing to be from an upslope HRU to one downslope HRU and/or stream segment, even if multiple cascade links are specified from an upslope HRU. The cascade link with the greatest fraction of contributing area (defined by parameter hru_pct_up, see Section 3.9.4.9 for details) in the upslope HRU is selected as the single outflow from the HRU.
- **cascade_tol:** Cascade area below which a cascade link is ignored. Contributing areas less than cascade_tol are evenly distributed to all other cascade paths originating in the upslope HRU. Unit = [acre]
- **precip_units:** The unit of all precipitation data.
- **temp_units:** The unit of all temperature data.
- **elev_units:** The unit of altitude/elevation data.
- **mxsziter:** Maximum number of iterations for which soil-zone states are computed

- **rad_conv:** Conversion factor to Langleys for observed radiation.
- **rpt_days:** Frequency that summary tables are written to the GSFLOW Water-Budget File (0 = none; >0 = frequency in days, e.g., 1 = daily, 7 = every 7th day).
- **soilzone_aet_flag:** If this box is checked, values of soil-water evapotranspiration (ET) are computed based on the potential evapotranspiration (PET) rate instead of the unsatisfied ET rate (PET rate less canopy, snowpack, impervious storage, and surface-depression storage ET).

3.9.4.6. HRU Index

Select *Models > GSFLOW > HRU Index* to assign HRU indices (or *HRU identification numbers*) to the model cells with the Data Editor. Each HRU is identified by a numerical index. Assignment of the index to the HRU is arbitrary, but indices must be unique, consecutive, and start with 1.

3.9.4.7. HRU Types

In GSFLOW, each HRU can be designated as one of the following four types: land, lake, swale, or inactive.

- **Land HRU:** A land HRU is the most common type of HRU; land HRUs simulate all supported hydrologic processes.
- **Lake HRU:** When the model domain includes large lakes, these areas should be delineated separately and designated as a lake-type HRU, where only precipitation, potential solar radiation, and lake evaporation are computed. Other processes such as plant canopy interception, surface runoff, and interflows through the soil zone are assumed zero and snowpack is computed as a liquid equivalent. **Lake HRU** is available only if the MODFLOW Lake Package is activated.
- **Swales** are land-type HRUs in which the surface runoff and interflow components of lateral flow are not present. These HRUs typically correspond to low areas in the model domain that capture flow and only contribute to groundwater flow.
- **Inactive HRUs** allow for areas within the model domain to be excluded from the simulation. No computations are performed for inactive HRUs.

To edit HRU Types, select *Models > GSFLOW > HRU Types* to open the HRU Data dialog box (Figure 131) that contains a table displaying the HRU Index and HRU Type. You may double-click a cell in the HRU Type

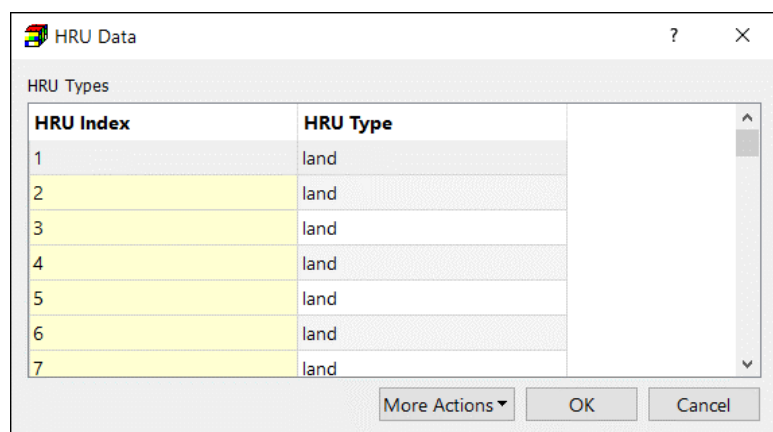


Figure 131. The HRU Data dialog box for editing HRU Types

column and then select another type from a dropdown box. To modify the values of multiple

HRUs, you may select them (by dragging over many rows or using combination of shift+click and ctrl+click), press the SPACE bar to display the dropdown box, and then select an item from the dropdown box.

3.9.4.8. HRU Lake ID

Select this menu item to open the HRU Data dialog box (Figure 132) that allows you to associate lake identification numbers (Lake ID) with HRUs. More than one HRU can be associated with each lake. Lake ID must exist in the Lake Package (Section 3.9.1.1.9).

This menu item is available only if the MODFLOW Lake Package is active and at least an HRU is Lake HRU.

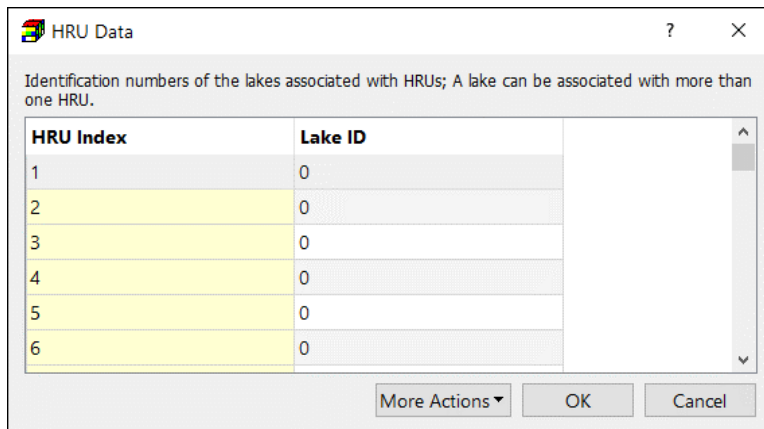


Figure 132. The HRU Data dialog box for assigning lake identification numbers to HRUs

3.9.4.9. Cascading Flow Links

GSFLOW uses a directed, acyclic-flow network (Ford, et al., 1956) in the cascading-flow procedure developed for routing surface runoff and interflow among HRUs and to streams and lakes. Flow paths start at the highest upslope HRUs and continue through downslope HRUs until reaching a stream segment or lake. Cascading flow may occur along many different paths and satisfies soil-zone storage of downslope HRUs before being added as inflow to stream segments or lakes. If the routing switch **cascade_flg** (in Section 3.9.4.5) is set to “allow many to many”, surface runoff and interflow from one upslope HRU may provide inflow to as many downslope HRUs, stream segments, and lakes as is required to conceptualize drainage in the watershed.

The cascading-flow procedure for routing surface runoff is simulated by one of the two Surface-Runoff Modules given in Section 3.9.4.11.7. The cascading interflow is simulated by the Soil-Zone Module (Section 3.9.4.11.5).

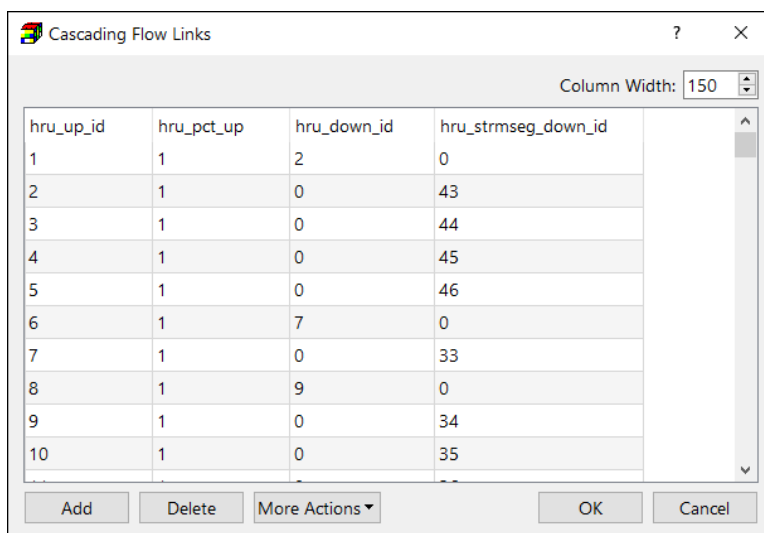


Figure 133. The Cascading Flow Links dialog box

The following four parameters are required by GSFLOW for calculating the flows between one HRU and another HRU or stream segment. They are entered in the Cascading Flow Links dialog box (Figure 133).

- **hru_up_id**: Identification number of the upslope HRU.
- **hru_down_id**: Identification number of the downslope HRU.
- **hru_pct_up**: Fraction (between 0 and 1) of outflow from upslope HRU to be added as inflow to a downslope HRU or stream segment. This fraction represents the ratio of the contributing area to the total area of the upslope HRU to be routed to a downslope HRU.
- **hru_strmseg_down_id**: Identification number of stream segment that receives inflow from an upslope HRU.

3.9.4.10. Ground Surface Elevation

Select *Models > GSFLOW > Ground Surface Elevation* to assign ground surface elevation to the model cells with the Data Editor. The elevation data should be expressed in the unit defined by `elev_units` in Section 3.9.4.5. The elevation data are used to determine the slope and aspect of the model cells with a “four-neighbors” method (Carter, 1992; Eytan, 1991). The average slope and aspect values of cells within an HRU are assigned as the slope and aspect to the HRU.

3.9.4.11. PRMS Processes and Modules

The PRMS simulation of the hydrologic cycle of a watershed is broken into several processes. Each of these processes represents either one of the major hydrological processes. A module is computer source code used to simulate a hydrologic process. A given process can have several modules in the library, each representing an alternative conceptualization or approach to simulation of the hydrologic process. Each module needs specific inputs to execute, and computes specific outputs that can be used as inputs to other modules. The PRMS processes are described in Appendix 1 of the PRMS-IV user guide (Markstrom, et al., 2015) and documents of subsequent updates that can be downloaded from the USGS website⁶³.

All PRMS processes are required for a GSFLOW simulation. To edit the data of a process, select the respective menu item. Most processes require several input parameters. For example, Figure 134 shows the required input parameters of the Interception process. You can click on an [Edit] button in the table to specify the values of the respective parameter. Most parameter values are specified in a tabular format – either one value per HRU or several values per HRU (e.g., monthly values per HRU). Once the parameter values are saved, the parameter name of that row is highlighted in a bold font.

⁶³ <https://www.usgs.gov/software/gsfLOW-coupled-groundwater-and-surface-water-flow-model>

Depending on whether required parameters are specified or not, the items of the Process menu may be decorated with the following icons:

- ✓ Indicates that all required parameters have been specified and will be used in the simulation.
- ✓ Indicates that a portion of required parameters has been specified. Default values of unspecified parameters are used in the simulation.
- No Icon** Indicates that none of the required parameters has been specified. Default values of all parameters are used in the simulation.

The following sections list the required parameters by process and module. For detailed description of the processes, refer to Appendix 1 of the PRMS-IV user guide (Markstrom, et al., 2015).

3.9.4.11.1. Interception

The Interception Process has a single module that computes volume of intercepted precipitation, evaporation from intercepted precipitation, and throughfall that reaches the soil or snowpack. The required parameters of the Interception module are entered in the Interception dialog box (Figure 134) and are described below.

- **cov_type [-]**: Vegetation cover type for each HRU.
- **covden_sum [decimal fraction]**: Summer vegetation cover density for the major vegetation type in each HRU.
- **covden_win [decimal fraction]**: Winter vegetation cover density for the major vegetation type in each HRU.
- **snow_intcp [inches]**: Snow interception storage capacity for the major vegetation type in each HRU.
- **srain_intcp [inches]**: Summer rain interception storage capacity for the major vegetation type in each HRU.
- **wrain_intcp [inches]**: Winter rain interception storage capacity for the major vegetation type in each HRU.

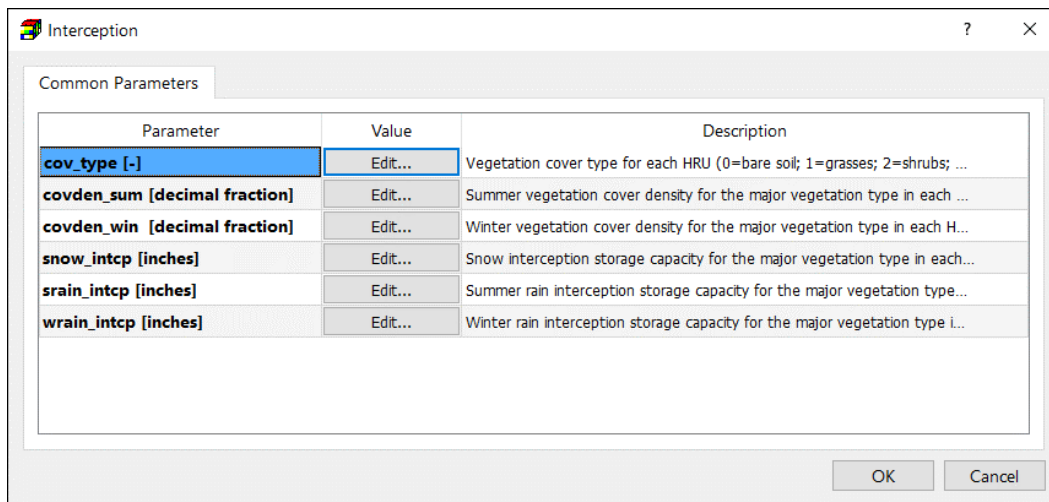


Figure 134. The Interception dialog box

- **wrain_intcp [inches]:** Winter rain interception storage capacity for the major vegetation type in each HRU.

3.9.4.11.2. Potential Evapotranspiration

The Potential Evapotranspiration process includes several modules that calculate the amount of potential evapotranspiration and determine the state of transpiration in HRUs. Figure 135 shows the dialog box for specifying the input parameters. The user selects one of the modules from the *Potential Evapotranspiration Module* dropdown box, and then enters the required parameters in the corresponding tab for the selected module. The parameters in the Common Parameters tab are used by all modules. The tabs and their associated parameters are as follows.

- **Common Parameters** tab includes parameters that are used by all modules.
 - **lake_evap_adj [decimal fraction]:** Monthly (January to December) adjustment factor for potential ET for each lake. This parameter is required when any Lake HRU exists.
- **potet_hamon** tab contains the data required for computing the potential evapotranspiration by using the Hamon formulation (Hamon, 1961).
 - **hamon_coef [-]:** Monthly (January to December) air temperature coefficient used in Hamon potential ET computations for each HRU.
- **potet_hs** tab contains the data required for computing the potential evapotranspiration for each HRU by using the Hargreaves-Samani formulation (Hargreaves, et al., 1982)
 - **hs_krs [decimal fraction]:** Monthly (January to December) adjustment factor used in Hargreaves-Samani potential ET computations for each HRU.

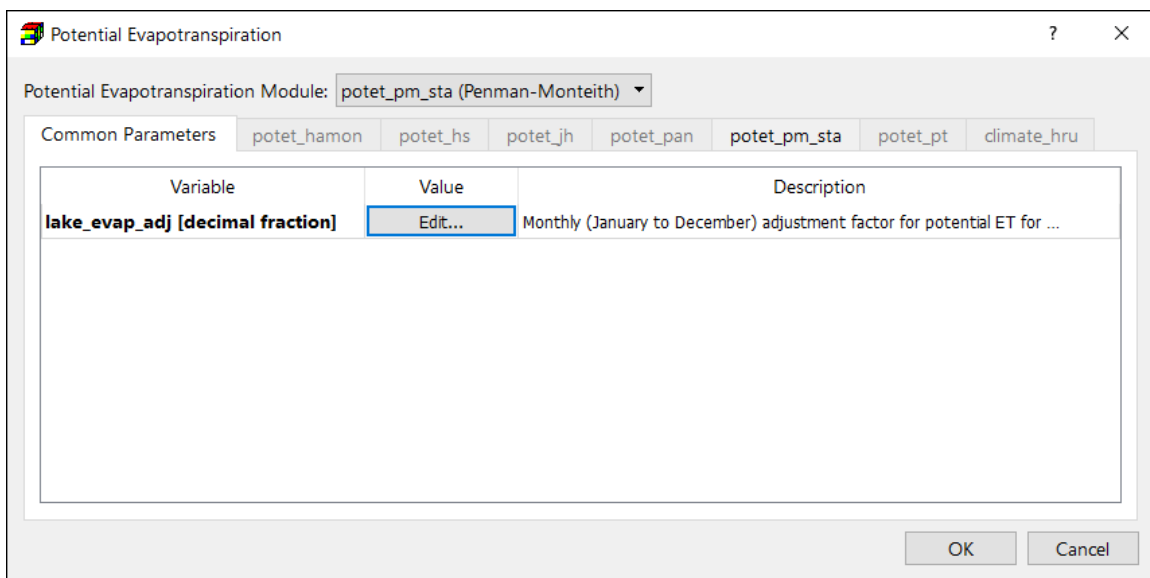


Figure 135. The Potential Evapotranspiration dialog box

- **potet_jh** tab contains the data required for computing the potential evapotranspiration by using the Jensen-Haise formulation (Jensen, et al., 1963)
 - **jh_coef [per degrees Fahrenheit]:** Monthly (January to December) air temperature coefficient used in Jensen-Haise potential ET computations for each HRU.
 - **jh_coef_hru [per degrees Fahrenheit]:** Air temperature coefficient used in Jensen-Haise potential ET computations for each HRU.
- **potet_pan** tab contains the data required for computing the potential evapotranspiration for each HRU by using pan-evaporation data.
 - **epan_coef [decimal fraction]:** Monthly (January to December) evaporation pan coefficient for each HRU.
 - **hru_pansta [-]:** The pan evaporation station used to compute HRU potential ET.
- **potet_pm_sta** tab contains the data required for computing the potential evapotranspiration by using the Penman-Monteith formulation (Penman, 1948; Monteith, 1965) using specified windspeed and humidity.
 - **crop_coef [decimal fraction]:** Monthly (January to December) crop coefficient for each HRU.
 - **hru_humidity_sta [-]:** The humidity measurement station for each HRU.
 - **hru_windspeed_sta [-]:** The wind speed measurement station for each HRU.
 - **pm_d_coef [seconds/meter]:** Monthly (January to December) Penman-Monteith potential ET D wind speed coefficient for each HRU.
 - **pm_n_coef [degrees Celsius per day]:** Monthly (January to December) Penman-Monteith potential ET N temperature coefficient for each HRU.
- **potet_pt** tab contains the data required for computing the potential evapotranspiration by using the Priestley-Taylor formulation (Priestley, et al., 1972).
 - **pt_alpha [decimal fraction]:** Monthly (January to December) adjustment factor used in Priestley-Taylor potential ET computations for each HRU.
 - **humidity_percent [percent]:** Monthly (January to December) humidity for each HRU.
- **climate_hru:** The Climate by HRU module (`climate_hru`) allows for input of pre-determined potential evapotranspiration to each HRU. This enables the user to use any method to pre-process and distribute climate data to HRUs outside of a GSFLOW simulation.
 - **hru_potetsta [-]:** The pre-determined potential evapotranspiration time-series are entered as observation values to observation points, which are associated with HRUs through the values of **hru_potetsta**.
 - **potet_cbh_adj [-]:** Monthly (January to December) adjustment factor to potential evapotranspiration for each HRU.

3.9.4.11.3. Precipitation Distribution

The Precipitation Distribution process includes several modules that interpolate/extrapolate precipitation measurements at one or more stations to HRUs. Figure 136 shows the dialog box for specifying the input parameters of the Precipitation Distribution process. The user selects one of the modules from the *Precipitation Distribution Module* dropdown box, and then enters the required parameters in the corresponding tab for the selected module. The parameters in the Common Parameters tab are used by all modules. The tabs and their associated parameters are described below.

- **Common Parameters** tab includes parameters that are used by all modules.
 - **adjmix_rain [-]**: Monthly (January to December) factor to adjust rain proportion in a mixed rain/snow event as defined by equation 6 of Markstrom, et al. (2008).
 - **tmax_allrain [temp_units]**: Monthly (January to December) maximum air temperature when precipitation is assumed to be rain; if HRU air temperature is greater than or equal to this value, precipitation is rain.
 - **tmax_allsnow [temp_units]**: Monthly (January to December) maximum air temperature when precipitation is assumed to be snow; if HRU air temperature is less than or equal to this value, precipitation is snow.
- **precip_1sta**: This option requires assigning a precipitation station to an HRU. A monthly correction factor, which accounts for elevation, spatial variation, topography, gage location, deficiencies in gage catch due to the effects of wind and other factors, is used to estimate the daily precipitation at the HRU according to equation 7 of Markstrom, et al. (2008).
 - **hru_psta [-]**: The precipitation observation point/station associated with each HRU.

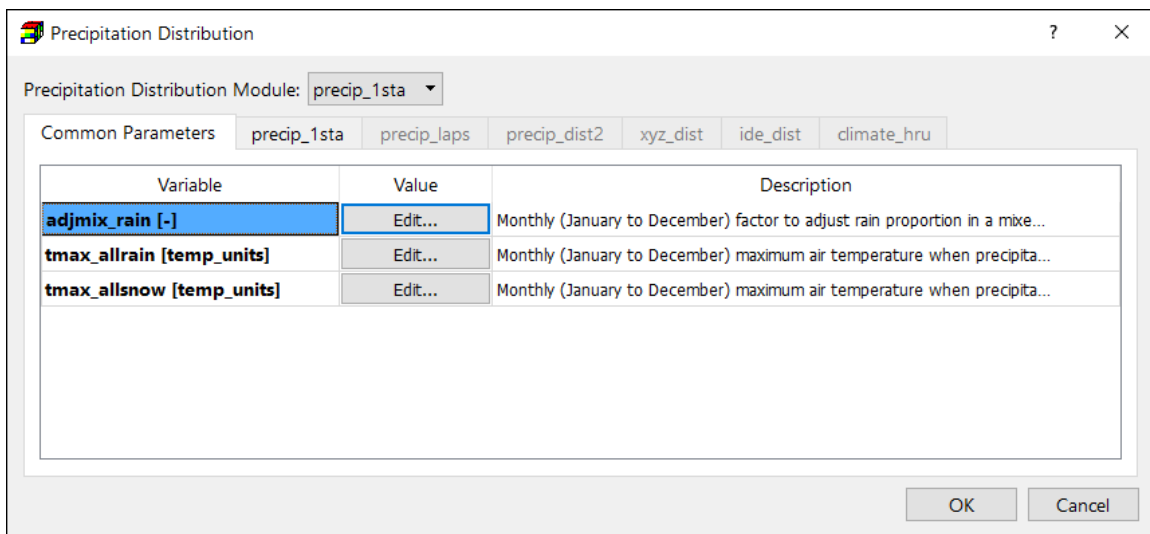


Figure 136. The Precipitation Distribution dialog box

- **rain_adj [-]:** Monthly (January to December) factor to adjust measured precipitation on each HRU to account for differences in elevation.
- **snow_adj [-]:** Monthly (January to December) factor to adjust measured precipitation on each HRU to account for differences in elevation.
- **precip_laps:** This module also uses equation 7 of Markstrom, et al. (2008), except that the correction factor is calculated for each HRU on the basis of computed lapse rates from two precipitation stations. The station that is most representative of precipitation on an HRU is designated as the base station. The station that is not the base station is called the lapse station and is used with the base station to calculate the lapse rate. The two stations assigned to each HRU are specified by `hru_psta` and `hru_plaps`.
 - **hru_plaps [-]:** The lapse precipitation measurement station used for lapse rate calculations for each HRU.
 - **hru_psta [-]:** The base precipitation measurement station used for lapse rate calculations for each HRU.
 - **padj_rn [precip_units]:** Monthly (January to December) factor to adjust rain lapse rate computed between station `hru_psta` and station `hru_plaps` (positive factors are used as multipliers and negative factors are made positive and substituted for the computed lapse rate).
 - **padj_sn [precip_units]:** Monthly (January to December) factor to adjust snow lapse rate computed between station `hru_psta` and station `hru_plaps` (positive factors are used as multipliers and negative factors are made positive and substituted for the computed lapse rate).
 - **pmn_mo [precip_units]:** Mean monthly (January to December) precipitation for each lapse precipitation measurement station.
- **precip_dist2:** This module weights measured precipitation from two or more stations by the inverse of the square of the distance between the centroid of an HRU and each station location.
 - **dist_max [feet]:** Maximum distance from an HRU to a measurement station for use in calculations.
 - **max_psta [-]:** Maximum number of precipitation measurement stations to distribute to an HRU.
 - **maxday_prec [precip_units]:** Maximum measured precipitation value above which precipitation is assumed to be in error.
 - **psta_mon [precip_units]:** Monthly (January to December) factor to precipitation at each measured station to adjust precipitation distributed to each HRU to account for differences in elevation.
 - **rain_mon [precip_units]:** Monthly (January to December) factor to rain on each HRU to adjust precipitation distributed to each HRU to account for differences in elevation.

- **snow_mon [precip_units]:** Monthly (January to December) factor to snow on each HRU to adjust precipitation distributed to each HRU to account for differences in elevation.
- **xyz_dist:** This option uses a three-dimensional, multiple-linear regression based on longitude, latitude, and elevation to distribute precipitation data from two or more measurement stations.
 - **adjust_rain [-]:** Monthly (January to December) factor used to adjust observed rain values for each precipitation station.
 - **adjust_snow [-]:** Monthly (January to December) factor used to adjust observed snow values for each precipitation station.
 - **ppt_add [precip_units] and ppt_div [precip_units]:** precipitation values that are used in the regression equation are transformed by adding ppt_add and the sum is divided by ppt_div.
 - **ppt_lapse [-]:** Monthly (January to December) precipitation lapse rate for each direction (X, Y, and Z).
 - **psta_freq_nuse [-]:** The subset of precipitation measurement stations used to determine if there is precipitation in the basin (0=station not used; 1=station used).
 - **psta_month_ppt [precip_units]:** Average monthly (January to December) precipitation at each precipitation measurement station.
 - **psta_nuse [-]:** The subset of precipitation measurement stations used in the distribution regression (0=station not used; 1=station used)
 - **rain_code [-]:** Monthly (January to December) flag setting the rule of how precipitation days are determined. Precipitation distribution is carried out only for days with precipitation. The following rules are used to determine a day with precipitation in the basin for the respective rain_code:
 - 1 = if psta_nuse stations have precipitation on that day;
 - 2 = if any precipitation station has precipitation on that day;
 - 3 = always;
 - 4 = if rain_day variable is set to 1 in a PRMS Data File;
 - 5 = if psta_freq_use stations have precipitation on that day.
 - **x_add [meters]:** Mean value for the climate station transformation equation for the longitude (X) coordinate.
 - **x_div [meters]:** Standard deviation for the climate station transformation equation for the longitude (X) coordinate (not 0.0).
 - **y_add [meters]:** Mean value for the climate station transformation equation for the longitude (Y) coordinate.
 - **y_div [meters]:** Standard deviation for the climate station transformation equation for the latitude (Y) coordinate.
 - **z_add [meters]:** Mean value for the climate station transformation equation for the elevation (Z) coordinate.

- **z_div [meters]:** Standard deviation for the climate station transformation equation for the elevation (Z) coordinate (not 0.0).
- **ide_dist:** This option can be used when data from three or more measurement stations are available to compute and distribute precipitation. The module uses a combination of Inverse Distance and Elevation (IDE) weighting for interpolating maximum and minimum precipitation-station data to each HRU.
 - **adjust_rain [-]:** Monthly (January to December) rain downscaling adjustment factor for each precipitation measurement station.
 - **adjust_snow [-]:** Monthly (January to December) snow downscaling adjustment factor for each precipitation measurement station.
 - **dist_exp [-]:** Exponent for inverse distance calculations.
 - **n_dist_psta [-]:** Number of precipitation measurement stations for inverse distance calculations.
 - **prcp_wght_dist [-]:** Monthly (January to December) precipitation weighting function for inverse distance calculations.
 - **psta_nuse [-]:** The subset of precipitation measurement stations used in the distribution regression (0=station not used; 1=station used).
- **climate_hru:** The option provides a method to input time-series of historic, current, and projected climate values, pre-distributed to each HRU.
 - **hru_psta [-]:** The precipitation measurement station associated with each HRU.
 - **rain_cbh_adj [-]:** Monthly (January to December) adjustment factor to measured precipitation determined to be rain on each HRU to account for differences in elevation or other factors.
 - **snow_cbh_adj [-]:** Monthly (January to December) adjustment factor to measured precipitation determined to be snow on each HRU to account for differences in elevation or other factors.

3.9.4.11.4. Snow Computation

The Snow Computation process simulates the initiation, accumulation, and depletion of a snowpack on each HRU. Snowpack dynamics are simulated through estimates of water and energy balances. These balances conserve mass and energy, such that the difference between inputs and outputs is equal to the change in snowpack storage.

The required parameters for the Snow Computation process are entered in the Snow Computation dialog box (Figure 137) and are described below.

- **albset_rna [decimal fraction]:** Fraction of rain in a mixed precipitation event above which the snow albedo is not reset; applied during the snowpack accumulation stage
- **albset_rnm [decimal fraction]:** Fraction of rain in a mixed precipitation event above which the snow albedo is not reset; applied during the snowpack melt stage.
- **albset_sna [inches]:** Minimum snowfall, in water equivalent, needed to reset snow albedo during the snowpack accumulation stage.

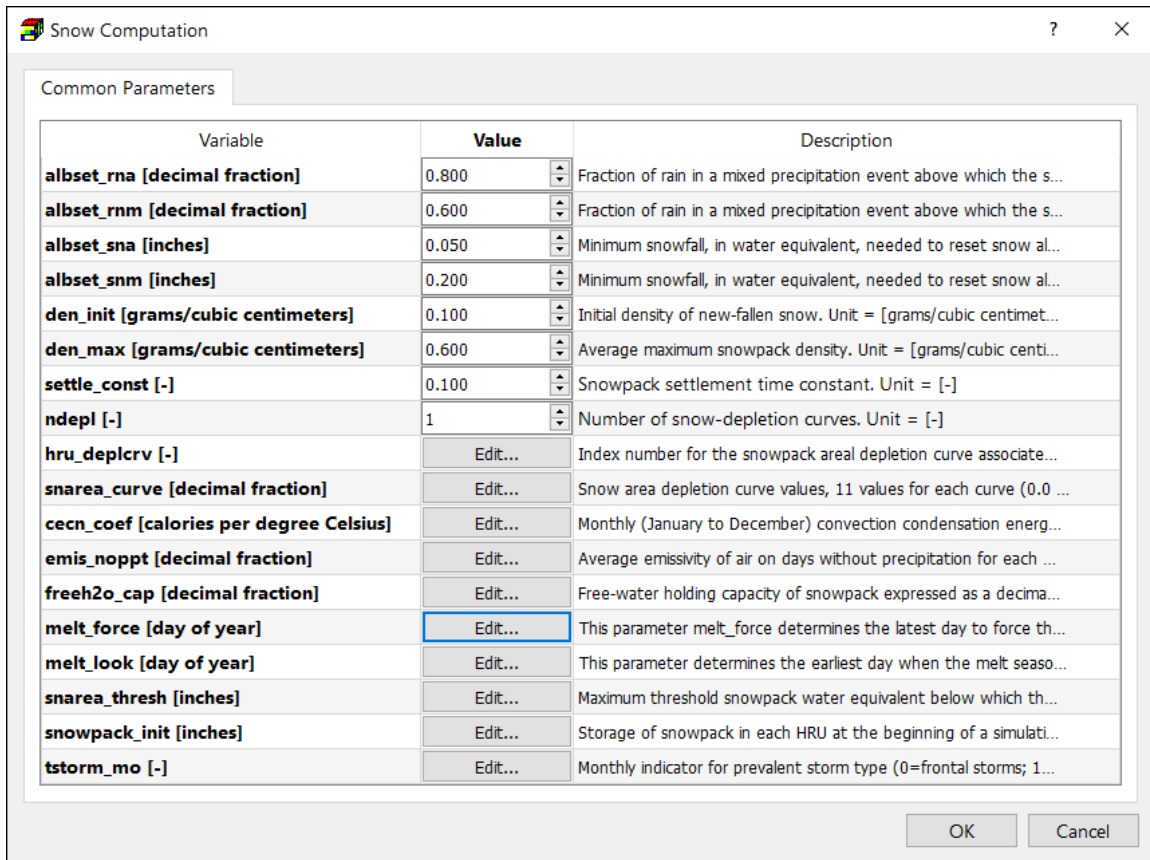


Figure 137. The Snow Computation dialog box

- **albset_snm [inches]:** Minimum snowfall, in water equivalent, needed to reset snow albedo during the snowpack melt stage.
- **den_init [grams/cubic centimeters]:** Initial density of new-fallen snow.
- **den_max [grams/cubic centimeters]:** Average maximum snowpack density.
- **settle_const [-]:** Snowpack settlement time constant.
- **ndepl [-]:** Number of snow-depletion curves.
- **hru_deplcrv [-]:** Index number for the snowpack areal depletion curve associated with each HRU. Unit = [-]
- **snarea_curve [decimal fraction]:** Snow area depletion curve values, 11 values for each curve (0.0 to 1.0 in 0.1 increments).
- **cecn_coef [calories per degree Celsius]:** Monthly (January to December) convection condensation energy coefficient for each HRU.
- **emis_noppt [decimal fraction]:** Average emissivity of air on days without precipitation for each HRU.
- **freeh2o_cap [decimal fraction]:** Free-water holding capacity of snowpack expressed as a decimal fraction of the frozen water content of the snowpack (pk_ice) for each HRU.
- **melt_force [day of year]:** This parameter determines the latest day to force the initiation of melt season.

- **melt_look [day of year]:** This parameter determines the earliest day when the melt season can begin.
- **snarea_thresh [inches]:** Maximum threshold snowpack water equivalent below which the snow-covered-area curve is applied.
- **snowpack_init [inches]:** Storage of snowpack in each HRU at the beginning of a simulation.
- **tstorm_mo [-]:** Monthly indicator for prevalent storm type (0=frontal storms; 1=convective storms) for each HRU

3.9.4.11.5. Soil Zone

The Soil-Zone process simulates the soil-zone hydrologic processes. Computation of the water content of the soil zone is based on the summation of all moisture depletions and accretions. Depletions include evapotranspiration, drainage to the groundwater reservoir, fast and slow interflow, and saturation excess surface runoff (herein called Dunnian surface runoff) (Dunne, et al., 1970). Accretions are soil infiltration as computed by the Surface Runoff Module and any cascading Dunnian surface runoff and interflow from upslope HRUs as computed by the Soil-Zone Module.

The required parameters for the Soil-Zone process are entered in the Soil Zone dialog box (Figure 138) and are described below.

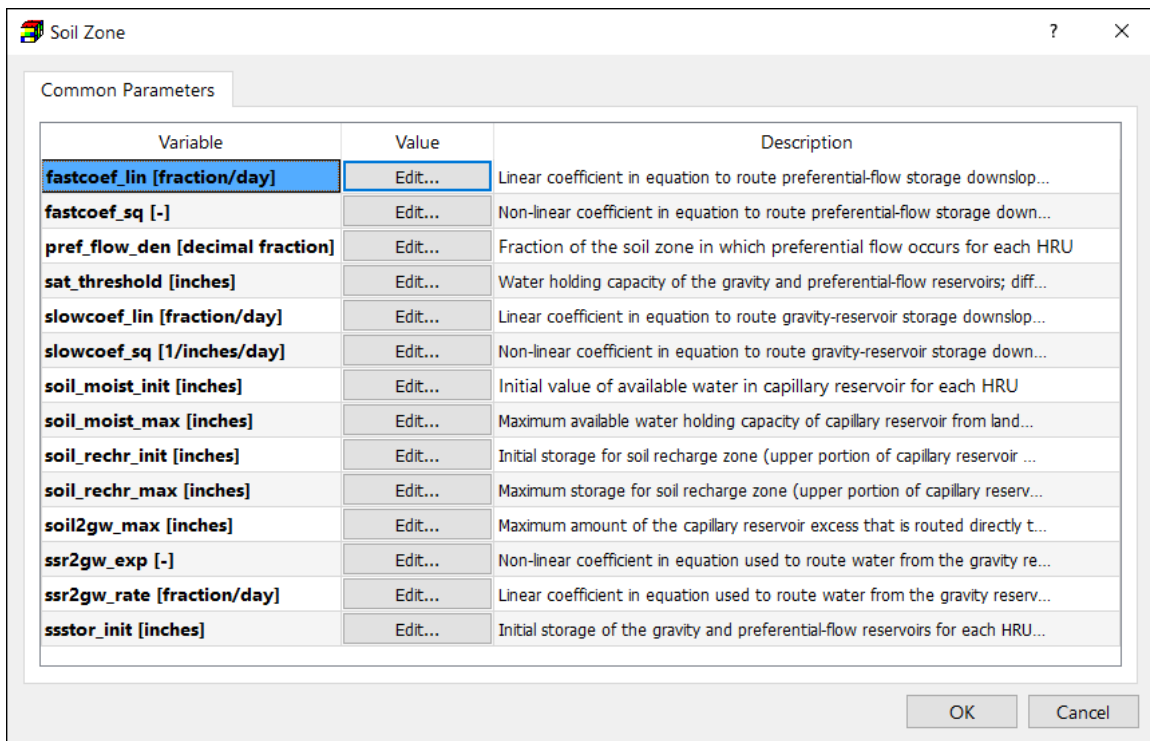


Figure 138. The Soil Zone dialog box

- **fastcoef_lin [fraction/day]:** Linear coefficient in equation to route preferential-flow storage downslope for each HRU
- **fastcoef_sq [-]:** Non-linear coefficient in equation to route preferential-flow storage downslope for each HRU
- **pref_flow_den [decimal fraction]:** Fraction of the soil zone in which preferential flow occurs for each HRU
- **sat_threshold [inches]:** Water holding capacity of the gravity and preferential-flow reservoirs; difference between field capacity and total soil saturation for each HRU
- **slowcoef_lin [fraction/day]:** Linear coefficient in equation to route gravity-reservoir storage downslope for each HRU
- **slowcoef_sq [1/inches/day]:** Non-linear coefficient in equation to route gravity-reservoir storage downslope for each HRU.
- **soil_moist_init [inches]:** Initial value of available water in capillary reservoir for each HRU
- **soil_moist_max [inches]:** Maximum available water holding capacity of capillary reservoir from land surface to rooting depth of the major vegetation type of each HRU.
- **soil_rechr_init [inches]:** Initial storage for soil recharge zone (upper portion of capillary reservoir where losses occur as both evaporation and transpiration) for each HRU; must be less than or equal to soil_moist_init.
- **soil_rechr_max [inches]:** Maximum storage for soil recharge zone (upper portion of capillary reservoir where losses occur as both evaporation and transpiration) for each HRU; must be less than or equal to soil_moist_max.
- **soil2gw_max [inches]:** Maximum amount of the capillary reservoir excess that is routed directly to groundwater for each HRU
- **ssr2gw_exp [-]:** Non-linear coefficient in equation used to route water from the gravity reservoirs to groundwater for each HRU.
- **ssr2gw_rate [inches/day]:** Linear coefficient in equation used to route water from the gravity reservoir to groundwater for each HRU.
- **ssstor_init [inches]:** Initial storage of the gravity and preferential-flow reservoirs for each HRU.

3.9.4.11.6. Solar Radiation

The Solar Radiation process includes algorithms (ddsolrad and ccsolrad) for estimating solar radiation at each HRU when measured solar-radiation data are not available. The Climate-by-HRU module can be used to input solar radiation directly. Figure 139 shows the dialog box for specifying the input parameters of Solar Radiation modules. The user must select a module from the *Solar Radiation Module* dropdown box, and then enters the required parameters in the corresponding tab for the selected module. The parameters in the Common Parameters tab are used by all modules. The tabs and their associated parameters are described below.

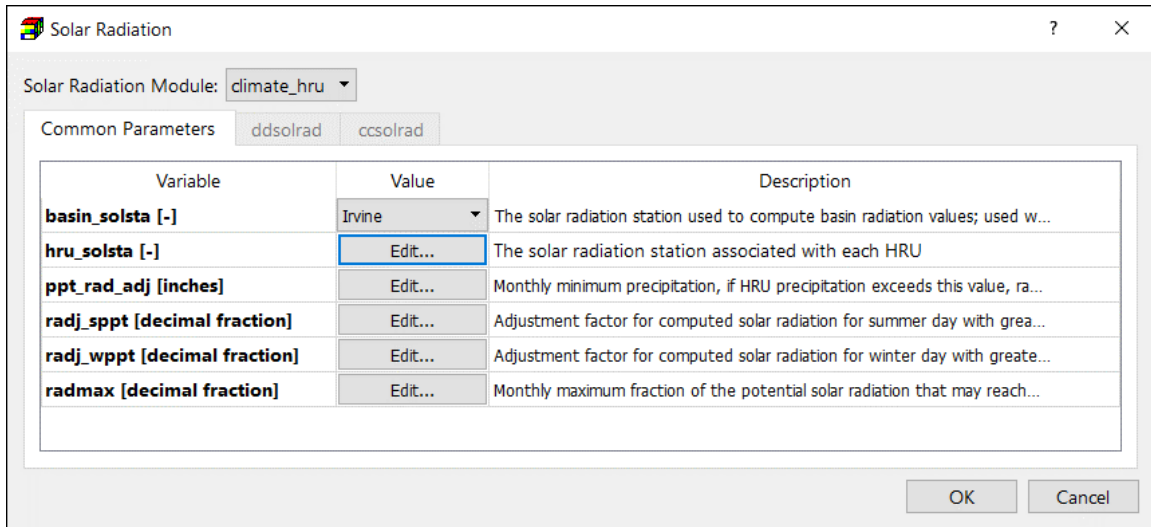


Figure 139. The Solar Radiation dialog box

- **Common Parameters** tab includes parameters that are used by all modules.
 - **basin_solsta [-]**: The solar radiation station used to compute basin radiation values; used when the number of solar-radiation measurement stations is greater than zero.
 - **hru_solsta [-]**: The solar radiation station associated with each HRU. The solar radiation measurements at those stations are used as the direct input values to the climate_hru module if it is selected.
 - **ppt_rad_adj [inches]**: Monthly minimum precipitation, if HRU precipitation exceeds this value, radiation is multiplied by radj_sppt or radj_wppt precipitation adjustment factor.
 - **radj_sppt [decimal fraction]**: Adjustment factor for computed solar radiation for summer day with greater than ppt_rad_adj inches precipitation for each HRU.
 - **radj_wppt [decimal fraction]**: Adjustment factor for computed solar radiation for winter day with greater than ppt_rad_adj inches precipitation for each HRU.
 - **radmax [decimal fraction]**: Monthly maximum fraction of the potential solar radiation that may reach the ground due to haze, dust, smog, and so forth, for each HRU.
- **ddsolrad**: This is the Degree-Day Solar-Radiation distribution module. It computes shortwave solar radiation with a modified degree-day method. This method was developed for the Rocky Mountain region of the United States. It is most applicable to regions where predominantly clear skies prevail on days without precipitation.
 - **dday_intcp [dday]**: Monthly (January to December) intercept in degree-day equation for each HRU.
 - **dday_slope [dday/temp_units]**: Monthly (January to December) slope in degree-day equation for each HRU.
 - **radadj_intcp [dday]**: Monthly (January to December) intercept in air temperature range adjustment to degree-day equation for each HRU.

- **radadj_slope [dday/temp_units]:** Monthly (January to December) slope in air temperature range adjustment to degree-day equation for each HRU.
- **tmax_index [temp_units]:** Maximum monthly (January to December) temperature used to determine precipitation adjustments to solar radiation for each HRU.
- **ccsolrad:** This is the Cloud-Cover Solar-Radiation distribution module. It computes solar radiation on each HRU, using a relation between sky cover and daily range in air temperature and a relation between solar radiation and sky cover. This procedure is applicable to more humid regions where extensive periods of cloud cover occur with and without precipitation. The cloud-cover method (Leavesley, et al., 1983) uses a relationship between solar radiation and sky cover developed by Thompson (1976) and a relationship between sky cover and a daily range in air temperature demonstrated by Tangborn (1978).
 - **ccov_intcp [-]:** Monthly (January to December) intercept in cloud-cover relationship.
 - **ccov_slope [-]:** Monthly (January to December) coefficient in cloud-cover relationship.
 - **crad_coef [-]:** Coefficient(B) in Thompson (1976) equation; varies by region, contour map of values in reference.
 - **crad_exp [-]:** Exponent(P) in Thompson (1976) equation.

3.9.4.11.7. Surface Runoff

The Surface-Runoff process includes two modules (srunoff_smidx and srunoff_carea) that compute surface runoff from infiltration excess and soil saturation by using a variable-source-area concept, where the runoff generating areas of the watershed surface vary in location and size over time. The srunoff_smidx module computes these values by using a non-linear, variable-source-area method, whereas the srunoff_carea module computes them by using a linear, variable-source-area method.

Rain throughfall, snowmelt, and any cascading Hortonian surface runoff from an upslope HRU are partitioned to the pervious, impervious, and surface-depression storage portions of each HRU on the basis of the fraction of impervious area (parameter *hru_percent_imperv*) and surface-depression storage area (parameter *dprst_area*) of the HRU. Both modules compute retention storage, evaporation, and runoff on impervious and depression storage areas of each HRU by using continuity. Both modules compute seepage from surface-depression storage. Surface runoff due to infiltration excess and exceeding impervious storage capacity are summed and referred to as Hortonian surface runoff (Horton, 1933).

Figure 140 shows the dialog box for specifying the input parameters of the Surface Runoff modules. The user must select a module from the *Surface Runoff Module* dropdown box, and then enters the required parameters in the tab of the selected module. To enable the surface depression calculation, check the *Calculate Surface Depression Storage (dprst_flag)* box and

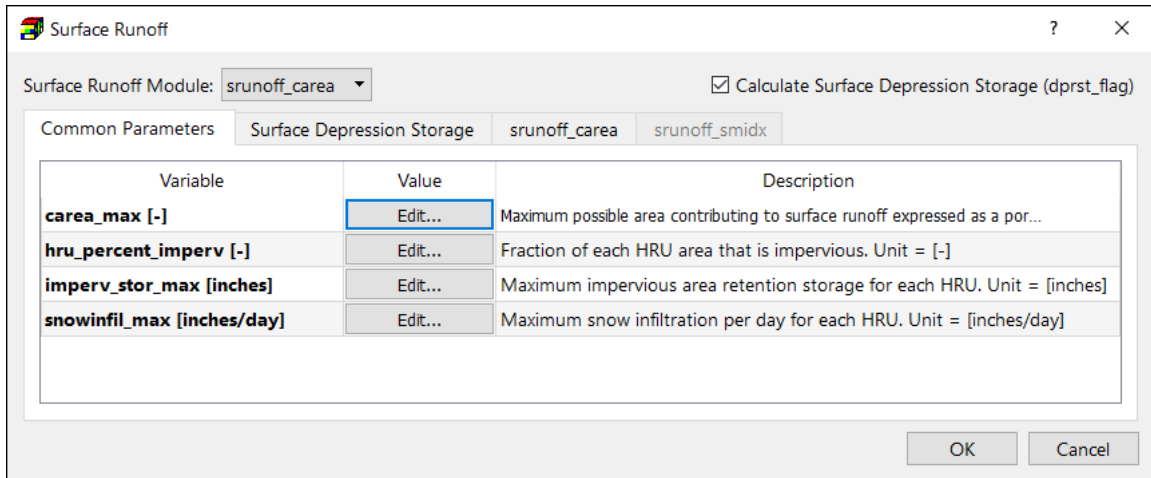


Figure 140. The Surface Runoff dialog box

enter the required parameters in the Surface Depression Storage tab. The parameters in the Common Parameters tab are used by all modules. The tabs and their associated parameters are described below.

- **Common Parameters** tab includes parameters that are used by all modules.
 - **carea_max [-]**: Maximum possible area contributing to surface runoff expressed as a portion of the HRU area.
 - **hru_percent_imperv [-]**: Fraction of each HRU area that is impervious.
 - **imperv_stor_max [inches]**: Maximum impervious area retention storage for each HRU.
 - **snowinfil_max [inches/day]**: Maximum snow infiltration per day for each HRU.
- **Surface Depression Storage**: The Surface-Runoff Modules can simulate surface-depression processes that account for the effect of numerous, small, unregulated water bodies. Although the effect of an individual surface depression may be negligible, numerous surface depressions can influence the hydrologic response of an HRU. Typically, surface depressions provide for water storage during and immediately after precipitation and snowmelt events; however, some may retain water year-round. A surface depression is distinct from a lake in that it is not large enough to warrant discretization as its own HRU. Examples of surface depressions include prairie potholes, farm and mill ponds, and stormwater-retention structures.
 - **dpsrt_area_option**: This option determines how the surface depress areas are defined.
 - **dprst_area [acres]**: Aggregate sum of surface-depression storage areas of each HRU.
 - **dprst_frac_hru [-]**: Fraction of each HRU area that has surface depressions.
 - **dprst_depth_avg [inches]**: Average depth of storage depressions at maximum storage capacity.

- **dprst_et_coef [-]:** Coefficient in linear flow routing equation for open surface depressions for each HRU.
- **dprst_flow_coef [1/day]:** Coefficient in linear flow routing equation for open surface depressions for each HRU.
- **dprst_frac_init [-]:** Fraction of maximum surface-depression storage that contains water at the start of a simulation.
- **dprst_frac_open [-]:** Fraction of open surface-depression storage area within an HRU that can generate surface runoff as a function of storage volume.
- **dprst_seep_rate_clos [1/day]:** Coefficient used in linear seepage flow equation for closed surface depressions for each HRU.
- **dprst_seep_rate_open [1/day]:** Coefficient used in linear seepage flow equation for open surface depressions for each HRU.
- **op_flow_thres [-]:** Fraction of open depression storage above which surface runoff occurs; any water above maximum open storage capacity spills as surface runoff.
- **sro_to_dprst_imperv [-]:** Fraction of impervious surface runoff that flows into surface-depression storage; the remainder flows to a stream network for each HRU.
- **sro_to_dprst [-]:** Fraction of pervious surface runoff that flows into surface-depression storage; the remainder flows to a stream network for each HRU.
- **va_clos_exp [-]:** Coefficient in the exponential equation relating maximum surface area to the fraction that closed depressions are full to compute current surface area for each HRU; 0.001 is an approximate rectangle; 1.0 is a triangle.
- **va_open_exp [-]:** Coefficient in the exponential equation relating maximum surface area to the fraction that open depressions are full to compute current surface area for each HRU; 0.001 is an approximate rectangle; 1.0 is a triangle.
- **srunoff_carea:**
 - **carea_min [-]:** Minimum possible area contributing to surface runoff expressed as a portion of the area for each HRU.
- **srunoff_smidx:**
 - **smidx_coef [-]:** Coefficient in non-linear contributing area algorithm for each HRU.
 - **smidx_exp [1/inch]:** Exponent in non-linear contributing area algorithm for each HRU.

3.9.4.11.8. Temperature Distribution

Air temperature is used in computations of evaporation, transpiration, sublimation, and snowmelt for each HRU. The Temperature-Distribution process include several modules to compute and distributes maximum, minimum, and average air temperatures in each HRU. The best option selection depends on the climate-distribution method that best represents the watershed and the number and locations of air-temperature measurement stations available.

Figure 141 shows the dialog box for specifying the input parameters of the Temperature Distribution modules. The user must select a module from the *Temperature Distribution Module* dropdown box, and then enters the required parameters in the tab of the selected module. The parameters in the Common Parameters tab are used by all modules. The tabs and their associated parameters are described below.

- **Common Parameters** tab includes parameters that are used by all modules.
 - **adjmix_rain [decimal fraction]:** Monthly (January to December) factor to adjust rain proportion in a mixed rain/snow event.
 - **tmax_allrain [temp_units]:** Monthly (January to December) maximum air temperature when precipitation is assumed to be rain; if HRU air temperature is greater than or equal to this value, precipitation is rain.
 - **tmax_allsnow [temp_units]:** Monthly (January to December) maximum air temperature when precipitation is assumed to be snow; if HRU air temperature is less than or equal to this value, precipitation is snow.
- **temp_1sta:** This module distributes air temperature to each HRU based on a measured value of maximum (tmax) and minimum (tmin) temperature and using monthly lapse-rate parameters (tmax_lapse and tmin_lapse) and temperature-adjustment parameters (tmax_adj and tmin_adj). The lapse rates represent the change in temperature with 1,000 units of elevation change.
 - **basin_tsta [-]:** The temperature station used to compute basin temperature values.
 - **max_missing [-]:** Maximum number of consecutive missing values allowed for any air temperature measurement station; missing value set to last valid value; 0=unlimited.
 - **hru_tsta [-]:** The base temperature station used for lapse rate calculations for each HRU.

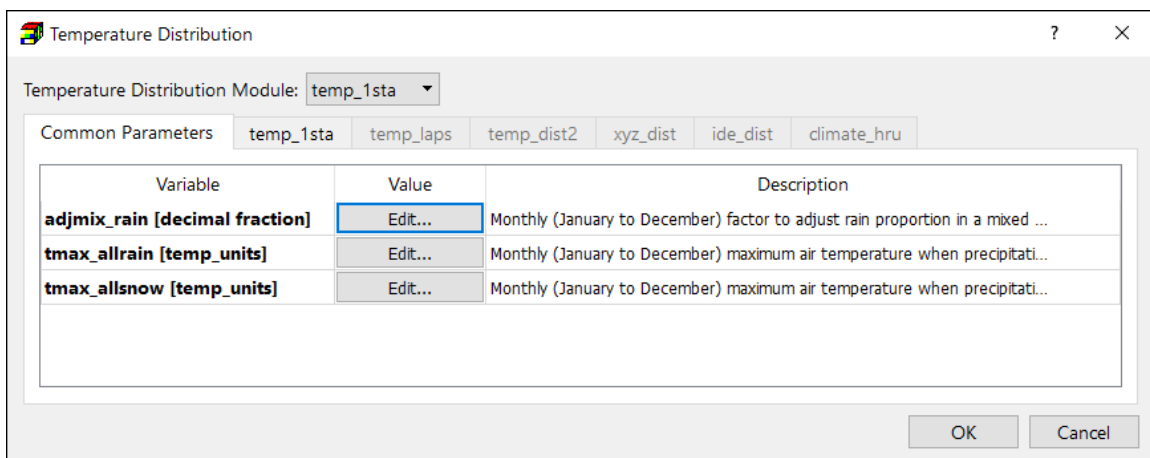


Figure 141. The Temperature Distribution dialog box

- **tmax_adj [temp_units]:** Adjustment to maximum air temperature for each HRU, estimated based on slope and aspect.
- **tmax_lapse [temp_units/elev_units]:** Monthly (January to December) values representing the change in maximum air temperature per 1,000 elev_units of elevation change for each HRU.
- **tmin_adj [temp_units]:** Adjustment to minimum air temperature for each HRU, estimated on the basis of slope and aspect.
- **tmin_lapse [temp_units/elev_units]:** Monthly (January to December) values representing the change in minimum air temperature per 1,000 elev_units of elevation change for each HRU.
- **temp_laps:** This module can be used when data from at least two measurement stations at different elevations are available to compute and distribute air temperature. Each HRU is assigned air temperatures based on daily computed lapse rates during each time step in the simulation from a pair of available measurement stations. The station that is most representative of the temperature at the HRU is designated as the base station and specified by parameter hru_tsta. The station that is not the base station is called the lapse station and specified by parameter hru_tlaps and is used with the base station to calculate the lapse rate.
 - **basin_tsta [-]:** The temperature station used to compute basin temperature values.
 - **max_missing [-]:** Maximum number of consecutive missing values allowed for any air temperature measurement station; missing value set to last valid value; 0=unlimited.
 - **hru_tlaps [-]:** The lapse temperature station used for lapse rate calculations for each HRU
 - **hru_tsta [-]:** The base temperature station used for lapse rate calculations.
 - **tmax_adj [temp_units]:** Adjustment to maximum air temperature for each HRU, estimated based on slope and aspect.
 - **tmin_adj [temp_units]:** Adjustment to minimum air temperature for each HRU, estimated based on slope and aspect.
- **temp_dist2:** This module distributes temperatures to the HRUs by computing weights based on lapse rates and the inverse of the square of the distance between the centroid of the HRU and the location of multiple air temperature measurement stations.
 - **basin_tsta [-]:** The temperature station used to compute basin temperature values.
 - **dist_max [feet]:** Maximum distance from an HRU to a measurement station for use in calculations.
 - **max_tsta [-]:** Maximum number of air temperature measurement stations to use for distributing temperature to any HRU.

- **lapsemax_max [temp_units/feet]:** Monthly (January to December) maximum lapse rate to constrain lowest maximum lapse rate based on historical daily air temperatures for all air temperature-measurement stations.
- **lapsemax_min [temp_units/feet]:** Monthly (January to December) maximum lapse rate to constrain lowest minimum lapse rate based on historical daily air temperatures for all air temperature-measurement stations.
- **lapsemin_max [temp_units/feet]:** Monthly (January to December) minimum lapse rate to constrain lowest maximum lapse rate based on historical daily air temperatures for all air temperature-measurement stations.
- **lapsemin_min [temp_units/feet]:** Monthly (January to December) minimum lapse rate to constrain lowest minimum lapse rate based on historical daily air temperatures for all air temperature-measurement stations.
- **monmax [temp_units]:** Monthly maximum air temperature to constrain lowest maximum air temperatures for bad values based on historical temperature for all measurement stations.
- **monmin [temp_units]:** Monthly minimum air temperature to constrain lowest maximum air temperatures for bad values based on historical temperature for all measurement stations.
- **tmax_adj [temp_units]:** Adjustment to maximum air temperature for each HRU, estimated based on slope and aspect.
- **tmin_adj [temp_units]:** Adjustment to minimum air temperature for each HRU, estimated based on slope and aspect.
- **xyz_dist:** This option uses a three-dimensional, multiple-linear regression based on longitude, latitude, and elevation to distribute temperature data from two or more measurement stations.
 - **solrad_elev [meters]:** Elevation of the solar radiation station used for the degree-day curves to distribute temperature [meters]
 - **max_lapse [temp_units/elev_units]:** Monthly (January to December) maximum air temperature lapse rate for each direction (X, Y, and Z).
 - **min_lapse [temp_units/elev_units]:** Monthly (January to December) minimum air temperature lapse rate for each direction (X, Y, and Z).
 - **tmax_add [temp_units]:** Mean value for the air-temperature measurement station transformation equation for maximum air temperature.
 - **tmax_div [temp_units]:** Standard deviation for the air-temperature-measurement station transformation equation for maximum air temperature (not 0.0).
 - **tmax_adj [temp_units]:** Adjustment to maximum air temperature for each HRU, estimated based on slope and aspect.
 - **tmax_allrain_dist [temp_units]:** Monthly (January to December) maximum air temperature when precipitation is assumed to be rain; if HRU air temperature is greater than or equal to this value, precipitation is rain.

- **tmax_allsnow_dist [temp_units]:** Maximum air temperature when precipitation is assumed to be snow; if mean air temperature is less than or equal to this value, precipitation is snow
- **tmin_add [temp_units]:** Mean value for the air-temperature-measurement station transformation equation for minimum air temperature.
- **tmin_div [temp_units]:** Standard deviation for the air-temperature-measurement station transformation equation for minimum air temperature (not 0.0).
- **tmin_adj [temp_units]:** Adjustment to minimum air temperature for each HRU, estimated on the basis of slope and aspect.
- **tsta_month_max [temp_units]:** Average monthly (January to December) maximum air temperature at each air-temperature-measurement station.
- **tsta_month_min [temp_units]:** Average monthly (January to December) minimum air temperature at each air-temperature-measurement station.
- **tsta_nuse [-]:** The subset of temperature stations used in the distribution regression (0=station not used; 1=station used)
- **x_add [meters]:** Mean value for the climate station transformation equation for the longitude (X) coordinate.
- **x_div [meters]:** Standard deviation for the climate station transformation equation for the longitude (X) coordinate (not 0.0).
- **y_add [meters]:** Mean value for the climate station transformation equation for the longitude (Y) coordinate.
- **y_div [meters]:** Standard deviation for the climate station transformation equation for the latitude (Y) coordinate.
- **z_add [meters]:** Mean value for the climate station transformation equation for the elevation (Z) coordinate.
- **z_div [meters]:** Standard deviation for the climate station transformation equation for the elevation (Z) coordinate (not 0.0).
- **ide_dist:** This option can be used when data from three or more measurement stations are available to compute and distribute temperature. The module uses a combination of Inverse Distance and Elevation (IDE) weighting for interpolating maximum and minimum temperature-station data to each HRU.
 - **dist_exp [-]:** Exponent for inverse distance calculations.
 - **n_dist_tsta [-]:** Number of air temperature measurement stations for inverse distance calculations. Unit = [-]
 - **solrad_elev [meters]:** Elevation of the solar radiation station used for the degree-day curves to distribute temperature.
 - **temp_wght_dist [-]** Monthly (January to December) temperature weighting function for inverse distance calculations.
 - **tmax_adj [temp_units]:** Adjustment to maximum air temperature for each HRU, estimated based on slope and aspect.
 - **tmax_allrain_sta [temp_units]:** Monthly (January to December) maximum air temperature when precipitation is assumed to be rain; if precipitation

measurement air temperature is greater than or equal to this value, precipitation is rain.

- **tmax_allsnow_sta [temp_units]:** Monthly (January to December) maximum air temperature when precipitation is assumed to be snow; if precipitation measurement air temperature is less than or equal to this value, precipitation is snow.
- **tmin_adj [temp_units]:** Adjustment to minimum air temperature for each HRU, estimated based on slope and aspect.
- **tsta_nuse [-]:** The subset of temperature stations used in the distribution regression (0=station not used; 1=station used).
- **climate_hru:** The option provides a method to input time-series of historic, current, and projected climate values, pre-distributed to each HRU.
 - **hru_tsta [-]:** The temperature measurement station associated with each HRU.
 - **tmax_cbh_adj [temp_units]:** Monthly (January to December) adjustment to maximum air temperature for each HRU, estimated based on slope and aspect.
 - **tmin_cbh_adj [temp_units]:** Monthly (January to December) adjustment to minimum air temperature for each HRU, estimated based on slope and aspect.

3.9.4.11.9. Transpiration Period

The Transpiration Period process include two modules that can be used to estimate the period of active transpiration for each HRU. The first module `transp_frost` determines whether the current time step is in a period of active transpiration by the killing frost method (Christiansen, et al., 2011). The second module `transp_tindex` uses a temperature index method.

Figure 142 shows the dialog box for specifying the input parameters of the modules. The user must select a module from the *Transpiration Period Module* dropdown box, and then enters the required parameters in the tab of the selected module. The parameters in the Common Parameters tab are used by all modules. The tabs and their associated parameters are described below.

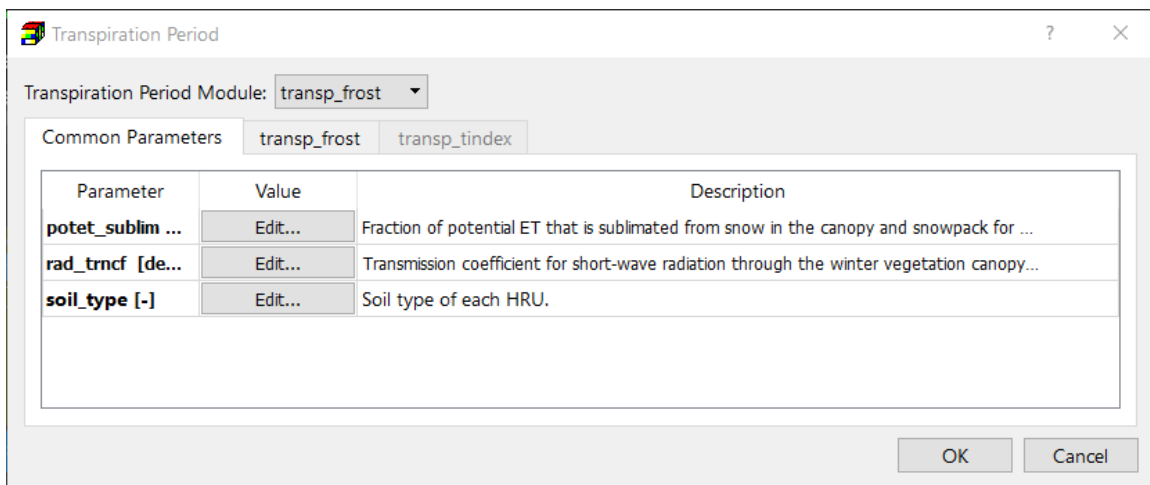


Figure 142. The Transpiration Period dialog box

- **Common Parameters** tab includes parameters that are used by all modules.
 - **potet_sublim [decimal fraction]:** Fraction of potential ET that is sublimated from snow in the canopy and snowpack for each HRU. Unit = [decimal fraction]
 - **rad_trncf [decimal fraction]:** Transmission coefficient for short-wave radiation through the winter vegetation canopy. Unit = [decimal fraction]
 - **soil_type [-]:** Soil type of each HRU.
- **transp_frost:** This option for determination of the period of active transpiration, or growing-season length, uses the killing frost approach. The growing-season length for each HRU can be defined as the time between the last (parameter spring_frost) and first (parameter fall_frost) freezing air temperatures.
 - **spring_frost [solar date]:** The solar date (number of days after winter solstice) of the last killing frost of the spring.
 - **fall_frost [solar date]:** The solar date (number of days after winter solstice) of the first killing frost of the fall.
- **transp_tindex:** This option for determination of the period of active transpiration uses a temperature index approach. This module computes a temperature index that is the cumulative sum of daily maximum temperature for each HRU after the model reaches the transpiration starting month (parameter transp_beg). The period of active transpiration for each HRU begins once the temperature index reaches a threshold (parameter transp_tmax). The period of transpiration for each HRU is terminated when the simulation reaches the month specified by parameter transp_end.
 - **transp_beg [month]:** Month to begin summing maximum air temperature for each HRU; when sum is greater than or equal to transp_tmax, transpiration begins.
 - **transp_end [month]:** Month to stop transpiration computations; transpiration is computed thru end of previous month.
 - **transp_tmax [temp_units]:** Temperature index to determine the start day of the transpiration period; the maximum air temperature for each HRU is summed starting with the first day of month transp_beg; when the sum exceeds this index, transpiration begins.

3.9.4.12. Output Control

The GSFLOW Output Control⁶⁴ dialog box (Figure 143) is used to define input parameters to the nhru_summary module (Regan, et al., 2017), which writes files of daily, monthly, and (or) mean monthly values of user-selected variables in a comma-separated values (CSV) format. These files are called Nhru Summary Files in GSFLOW. Each of these files has the number of values per line equal to the number of HRUs. The CSV format allows simulation results to be directly input to other software, such as statistical, graphical, and simulation models.

⁶⁴ Use *Models > Modflow > Output Control* to define the output options of MODFLOW.

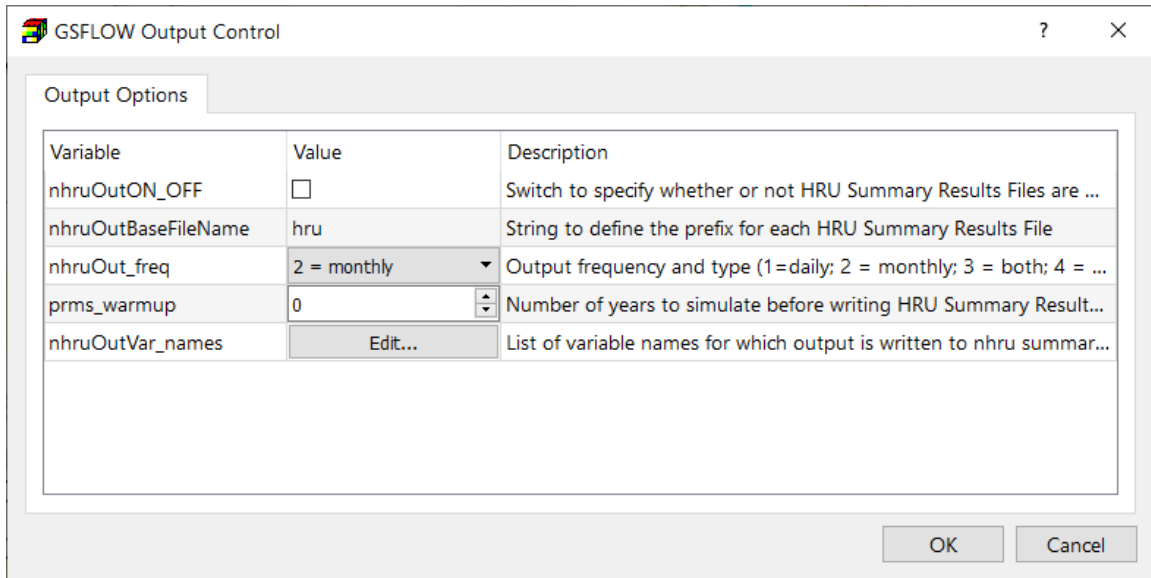


Figure 143. The GSFLOW Output Control dialog box

Any number of variables that are dimensioned by the value of nhru can be output. A separate file is generated for each selected variable. The module provides for specification of a warmup time period (parameter prms_warmup) to allow the simulation results to reach equilibrium between applied climatic conditions and dynamic surface hydrologic processes prior to writing to NHRU Summary Files.

The features of the GSFLOW Output Control dialog box are as follows.

- **nhruOutON_OFF:** Switch to specify whether or not HRU Summary Results Files are generated.
- **nhruOutBaseFileName:** Text string to define the prefix for each HRU Summary Results File.
- **nhruOut_freq:** Output frequency and type (1=daily; 2 = monthly; 3 = both; 4 = mean monthly).
- **prms_warmup:** Number of years to simulate before writing HRU Summary Results Files
- **nhruOutVar_names:** Click the [Edit] button to select variable name(s) for which output is written to nhru summary Comma Separated Values (CSV) output files(s). The results for each selected variable are written to a separate file. The filename is composed of nhruOutBaseFileName and the selected variable. For example, if nhruOutBaseFileName is "hru_" and the selected variable is "recharge", the filename is "hru_recharge.csv".


3.9.4.13. Inspect Model Data

Select this to check the geometry of the MODFLOW model and the consistency of the model data. Table 2 shows a list of items that are inspected by PM. The inspection report is saved in the *modflow_inspection_log.txt* file

3.9.4.14. Run

Select this menu item to open the Run Gsflow dialog box (Figure 144). The dialog box can be used to start the flow simulation with GSFLOW, to build input files to GSFLOW/MODFLOW/PRMS, or to check the model data. The available settings of the dialog box are described below.

1. Table

- **The build box** . Prior to running a simulation, PM uses the user-specified data to build input files for GSFLOW, MODFLOW, and PRMS. An input file to MODFLOW will be built if its build box is checked. If an input file does not exist or the input data has been modified since the last build, its build box will be checked automatically. All input files to PRMS and GSFLOW are always built.
- **Package** is the name of the package of the MODFLOW model.
- **Filename** is the name of the input file.
- **Status** shows the status (such as Ready, Not Exist, etc.) of the input file.

2. More Actions

- **Inspect Model Data:** Select this action to check the geometry of the MODFLOW model and the consistency of the model data. Table 2 shows a list of items that are inspected by PM. The inspection report is saved in the *modflow_inspection_log.txt* file.
- **Build Input Files:** Select this to build the input files with the checked build boxes. The name file modflow.nam and input file modflow.lmt for LMT8 package will also be built. The build log is stored in the *modflow_build_log.txt* file located in the model data folder.
- **Rebuild All Input Files:** Select this to rebuild all input files, except for existing read-only input files. The build log is stored in the *modflow_build_log.txt* file.

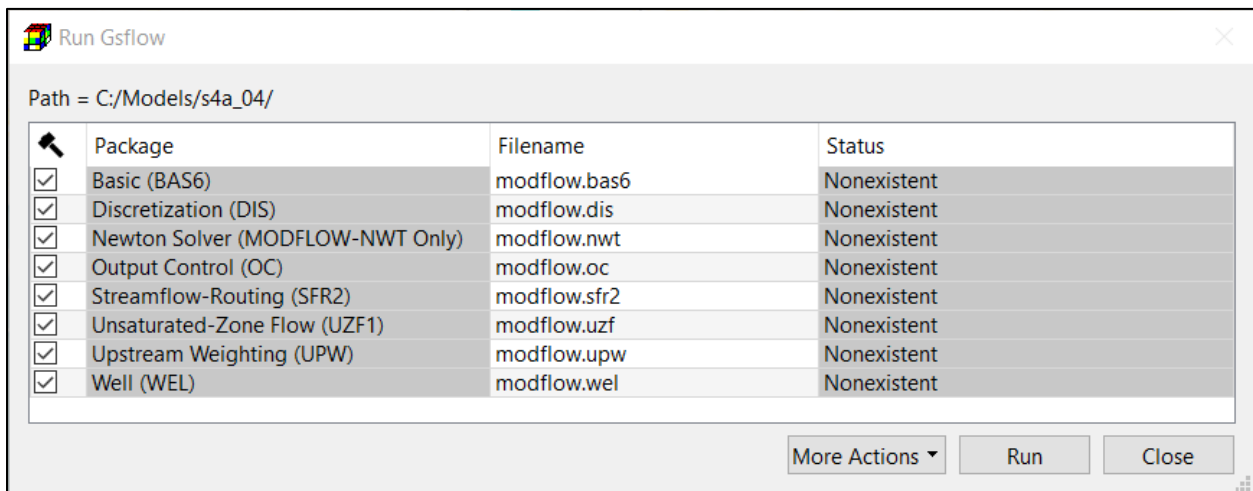


Figure 144. The Run Gsflow dialog box

- 3. Run:** Click this button to build input files and the start the simulation with GSFLOW. The GSFLOW run progress will be displayed in the GSFLOW Output window. You may terminate the simulation at any time by clicking on the Terminate button.

3.9.4.15. View

3.9.4.15.1. MODFLOW Run Listing File

Select this menu item to use the Text Editor (see Section 3.5.12) to display the run listing file `modflow.list`.

3.9.4.15.2. GSFLOW Log File

Select this menu item to use the Text Editor (see Section 3.5.12) to display the GSFLOW log file `gsflow.log`.

3.9.4.15.3. PRMS Water Budget File

Select this menu item to use the Text Editor (see Section 3.5.12) to display the surface water model water budget file `gsflow.out.csv`.

3.9.5. PEST

This menu provides the interface to PEST (Doherty, 2016), BeoPEST (Hunt, et al., 2010), PEST-GLM (Welter, et al., 2015; White, 2018; White, et al., 2020), and PESTPP-IES (White, et al., 2020). BeoPEST is a version of PEST that can run simulations on a parallel computing environment for the calibration of a deterministic model. PESTPP-IES is a version of PEST that works on a parallel computing environment to generate calibrated realizations. These codes are collectively referred to as PEST hereafter.

3.9.5.1. Prepare for Model Calibration with PEST

This section outlines the steps required to prepare input data for model calibration using PEST. During calibration, PEST adjusts parameter values to minimize the objective function, which is the sum of weighted squared residuals between model-calculated and observed values.

For this process to function correctly, multiple data files are required, including:

- Model input files
- Files for model pre- and post-processors
- A PEST control file, which contains run settings, adjustable parameters, and observation data

PM can be used to manage input data and prepare the required files at each of the major steps outlined in Figure 145 and described below.

1. Prepare Model Inputs

Before initiating the model calibration process, ensure that the model setup is complete and functioning correctly.

2. Define Calibration Targets

Calibration targets are selected observation data used to evaluate and enhance the fit of a numerical model to the real-world system being simulated. The primary objective of calibration in PEST is to adjust model parameters to minimize the difference between simulated values and their corresponding calibration targets.

To enter observation data and select calibration targets, use the Observation Tool, as described in Section 3.6.3. PM will automatically incorporate the selected calibration targets into the PEST control file before running PEST.

3. Define Adjustable Parameters

Parameterization in PEST involves defining the parameters that will be adjusted during calibration to improve the model's fit to observed data. The adjustable aquifer parameters depend on the selection of:

- The groundwater flow package
- The layer types
- The parameterization method

Table 3 lists the adjustable parameters supported by PM along with their corresponding parameter name prefixes. Adjustable parameters can be defined using two approaches:

- **Define adjustable parameters by zones**

In this approach, a model layer is subdivided into zones. A single parameter value is estimated for all model cells within each zone. PEST iteratively adjusts the zone parameters to improve the model's fit.

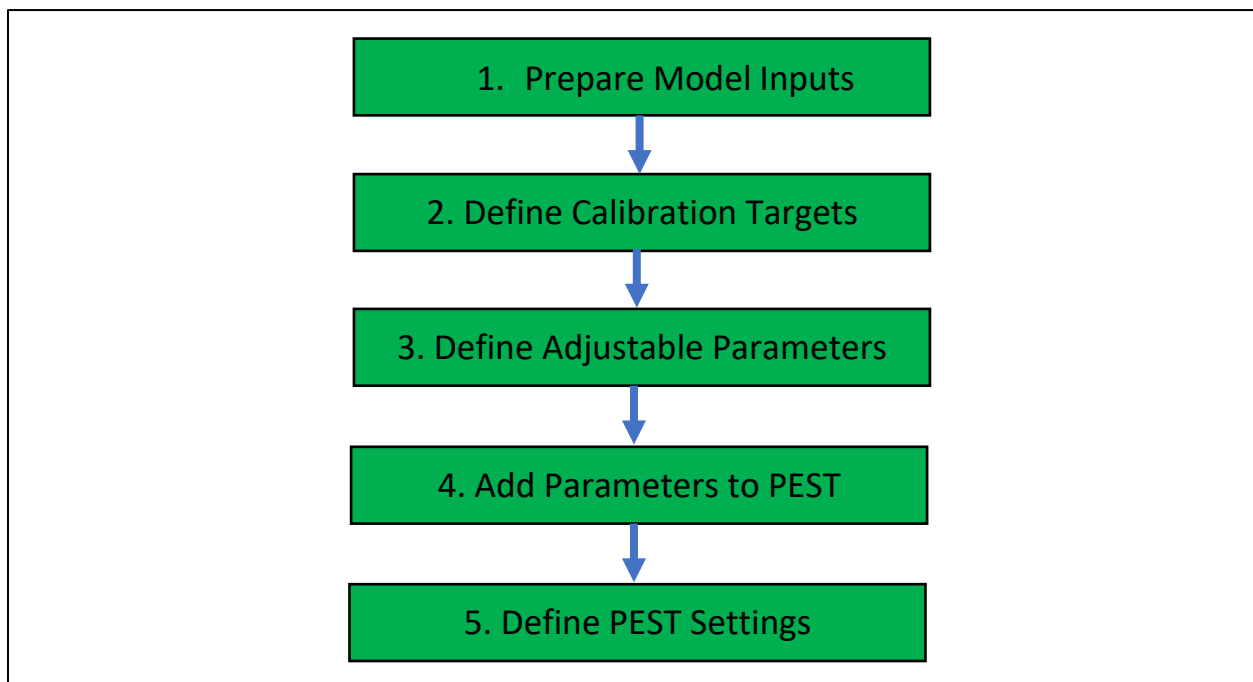


Figure 145. Steps to prepare files for model calibration with PEST

Steps:

- Select a parameter from the Parameters menu or select a package from *Models > MODFLOW > Packages and* navigate to the model layer where adjustable parameters will be defined.
- Assign a non-zero initial parameter value and a positive Parameter/Zone Number to the cells where a uniform parameter value should be estimated.
 - The Parameter/Zone Number must be unique within a parameter type (e.g., T, S, or Ss).
 - It must be an integer between 1 and 10,000.
- **Define adjustable parameters by pilot points.**

PM implements the Pilot Points Method, combined with Kriging, on a layer-by-layer basis. In this approach:

- Pilot points are assigned to specific aquifer parameters within a model layer.
- Initial parameter values are assigned to pilot points.
- The Kriging method interpolates the pilot point values to model cells during a calibration process.
- PEST iteratively adjusts the pilot point parameters to improve the model's fit.

Steps:

- Select the aquifer parameter (e.g., HK, VK, Ss, Sy) from the Parameters menu, then move to the desired model layer.
- Add pilot points to the model layer and assign initial parameter values to them.
- Assign negative Parameter/Zone Numbers **Error! Bookmark not defined.** to the cells where parameter values are estimated by pilot points.
 - The absolute values of these numbers are used as Zone numbers in the Pilot Points method.
 - Kriging within each zone is based on:
 - The pilot points in that zone
 - Its geostatistical structure
 - The associated variograms
- Define the geostatistical structure and variograms for each zone using the steps outlined in Section 3.4.2.4.5.
- Select the appropriate options in the Options tab (Figure 31).

Table 3. Adjustable Parameters through PEST within PM

Package	Parameter	Adjustable Parameter by Zones	Adjustable Parameter by Pilot Points
BCF	Confined layer: T, S, and VCONT ⁶⁵ Unconfined layer: HK, Sy, and VCONT ⁶⁵ Limited convertible layer: T, S, Sy, and VCONT ⁶⁵ Full convertible layer: HK, S, Sy, and VCONT ⁶⁵	Yes	Yes
LPF	All layer types: HK, HANI, VANI, VK, Ss, and Sy.	Yes	Yes
UPW	All layer types: HK, HANI, VANI, VK, Ss, and Sy.	Yes	Yes
DRN	Drain conductance: DRN	Yes	No
DRT1	Drain conductance: DRT	Yes	No
EVT	Maximum ET Flux: EVT	Yes	No
ETS1	Maximum ET Flux: ETS	Yes	No
FHB1	Multiplier to specified flow: FHBF Addend to specified head: FHBH	Yes	No
GHB	Boundary conductance: GHB	Yes	No
HBF6	Hydraulic characteristic: HFB	Yes	No
IBS	Inelastic Storage Factor: IBS	Yes	No
RCH	Multiplier for layer-wise recharge flux array: RCHM Recharge flux: RCH	Yes	No
RIV	Riverbed conductance: RIV	Yes	No
RES1	Vertical hydraulic conductivity of reservoir bed: RES	Yes	No
STR	Streambed conductance: STR	Yes	No
SFR2	Hydraulic conductivity of streambed: SFRHC	Yes	No
UZF1	Infiltration rate: UZF	Yes	No
WEL	Flow rate: WEL	Yes	No

4. Add Parameters to PEST

Once adjustable parameters are defined, they must be added to PEST to be used in the calibration process:

Steps:

- Open the PEST Settings dialog box (Figure 146) by selecting *Models > PEST > Settings*.
- In the Parameters tab, all defined adjustable parameters are listed.
 - Each row in the Parameters table represents either:
 - A zone-based parameter, or
 - A group of pilot points for an aquifer parameter in a model layer.
 - If opening the dialog for the first time, parameters will be automatically populated.
 - If additional parameters need to be added, select *More Actions > Search for Parameters*

⁶⁵ VCONT is used only if the layer is not the bottom layer.

- To define an adjustable parameter (e.g., HK, VK, Ss, Sy, etc.) for a model layer, select the desired parameter from the Parameters menu and move to the model layer.
- Add pilot points to the model layer and assign initial parameter values to the pilot points.
- Assign negative *Parameter/Zone Numbers* **Error! Bookmark not defined.** to the cells where the parameter values are to be estimated by the pilot points. The absolute values of the *Parameter/Zone Numbers* are used as the Zone numbers for the Pilot Points method. Kriging within each zone is based on the pilot points within the zone and its geostatistical structure and the associated variograms. You need to define the geostatistical structure and the associated variograms for each zone by using the procedure outlined in Section 3.4.2.4.5. You also need to select the appropriate options in the Options tab of Figure 31.

Once adjustable parameters are defined, you can add them to the Parameters tab of the PEST Settings dialog box (Figure 146) by selecting *Model > PEST > Settings*. Each row of the Parameters table represents a zone-wise parameter or pilot points of an aquifer parameter in a model layer. If the PEST Settings dialog box is opened the first time, the defined parameters will be listed in the Parameters Tab. Otherwise, select *More Actions > Search for Parameters* in that dialog box to add new parameters.

5. Define PEST Settings

PEST Settings refer to the configuration and control parameters specified in a PEST Control file (pest.pst). Most of the required data can be entered through the PEST Settings dialog box (Figure 146). The PEST Control file is a crucial component of model calibration, defining how the process is conducted through the key sections listed below. PM automates the management of these settings to streamline the calibration workflow. For a detailed explanation of each setting, refer to Section 3.9.5.2.

- **Control Data**

Defines general settings that control PEST's behavior during the calibration process.

- **Parameter Groups**

- Organizes adjustable parameters into logical groups based on their characteristics or the physical processes they represent.
- Provides settings that control finite-difference derivatives calculations for each group.

- **Parameter Data**

Contains settings for adjustable parameters, including:

- Parameter names
- Initial values
- Lower and upper bounds

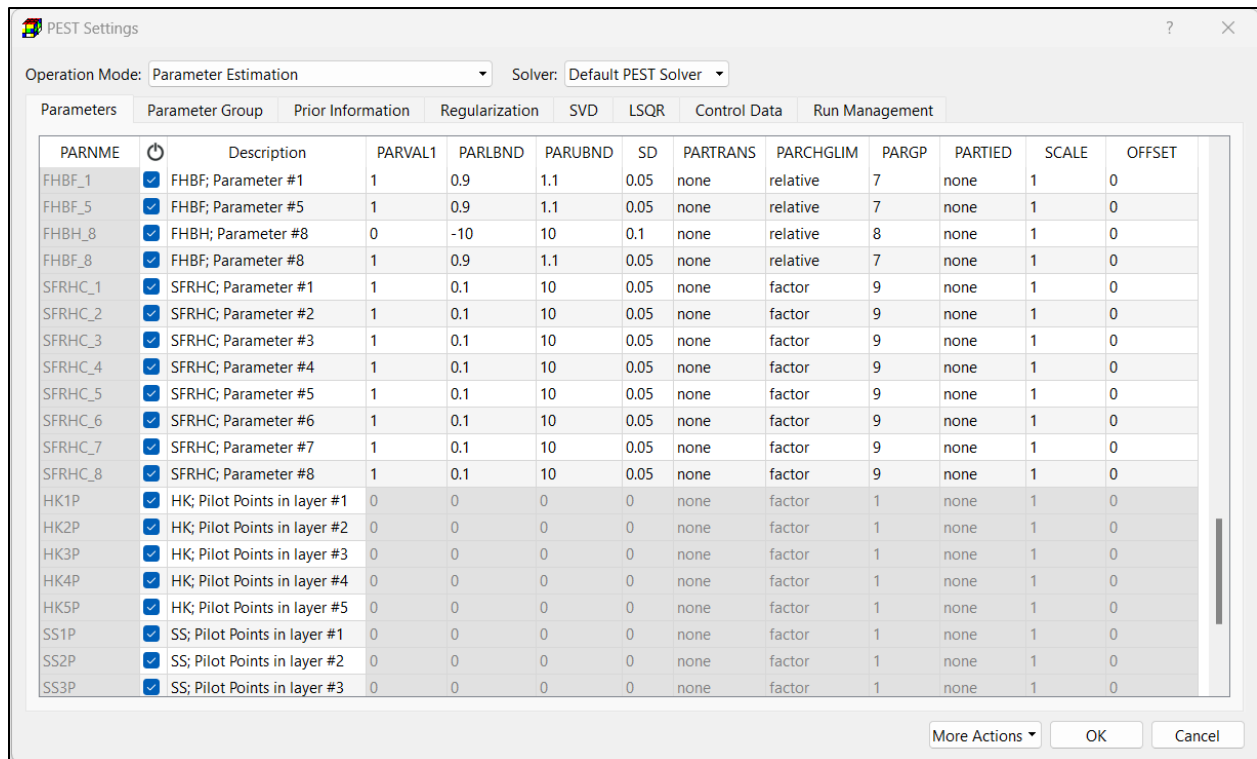


Figure 146. The Parameters tab of the PEST Settings dialog box

- Assigned parameter groups
- **Observation Groups**

Allows observations to be categorized into groups, facilitating the weighting and prioritization of different observation types.
- **Observation Data**

Contains calibration targets and their corresponding weights, which guide PEST's parameter adjustments to improve model fit.
- **Model Command Line**

This section specifies the commands that PEST must execute during each calibration iteration. PM uses the command "modelrun.bat", a Windows batch file that automates the following steps:

 - Preprocessing:
 - Executes RunFAC2REAL.bat to perform Kriging interpolation from pilot points to model cells.
 - Executes RunReal2Matrix.bat to update model input matrix files (e.g., HK or VK) based on Kriging results.
 - If applicable, FHBAdjust.exe updates input files for the FHB package, applying updated multipliers for specified-flow boundaries, and adds adds for specified-head boundaries.

- Groundwater Flow Simulation
 - MODFLOW or GSFLOW is called to perform flow simulation.
- Postprocessing:
 - mod2obs.exe and layerweight.exe extract simulated head values at observation wells and store them in an output file.
 - streamgage.exe extracts simulated streamflow values and store them in an output file.
 - lakegage.exe extracts simulated lake stage values and store them in an output file.

- **Model Input/Output**

This section defines the relationship between PEST template files, model input files, PEST instruction files, and model output files.

Each line in this section contains either a template file and its corresponding model input file, or an instruction file and its corresponding model output file.

- Template Files:

A template file is a replica of a model input file, except that:

 - Each parameter location is replaced with a unique character sequence.
 - During calibration, PEST replaces these sequences with updated parameter values.
 - The modified file is then written as the actual model input file.
- Instruction Files:

An instruction file contains directives that tell PEST how to extract data from the model output file.

 - Examples:
 - pest.headinstruction instructs PEST on reading pest.headoutput, which is generated by layerweight.exe.
 - pest.streaminstruction instructs PEST on reading pest.streamoutput, which is generated by streamgage.exe.

- **Prior Information**

- Prior information is treated as observations that apply directly to parameters.
- It serves as additional constraints in the calibration process, improving parameter estimation.

- **PESTPP-IES Run Management**

Additional PESTPP-IES control variables can be included in the PEST control file.

- PESTPP-IES control variables always start with ++.
- Refer to the PEST++ manual⁶⁶ for a complete list of available control variables.

⁶⁶ PEST++ and its documentation can be found in <https://github.com/usgs/pestpp/tree/master>

- Example:
 - ++ies_num_reals(100) instructs PESTPP-IES to generate 100 realizations for uncertainty analysis.

3.9.5.2. PEST Settings

The required inputs and options for running PEST are specified in the PEST Settings dialog box (Figure 146), which includes several tabs and a few options. The variable names used in this dialog box are identical as those given in the PEST Manual (Doherty, 2016) and the PEST++ manual **Error! Bookmark not defined.** The available options are described below. The tabs are described in the next subsections.

1. Operation Mode

- **Parameter Estimation:** PEST will run in the “estimation” mode and will use the available information to estimate parameters defined in the Parameters Tab by running the model as many times as needed. Solution of the inverse problem can be based on the Gauss-Marquardt-Levenberg method. Alternatively, singular value decomposition or LSQR can be used to solve the inverse problem. When many parameters require estimation, LSQR is much faster than that of singular value decomposition.
- **Sensitivity Analysis:** When this option is selected, the maximum number of optimization iterations (see NOPTMAX in the Control Data tab) will be set to -1. PEST will be used and will run in the “estimation” mode but will terminate execution immediately after it has calculated the Jacobian matrix for the first time. The parameter covariance, correlation coefficient and eigenvector matrices will be written to the run record file, and parameter sensitivities will be written to the sensitivity file; these are based on the initial parameter set defined either by Pilot Points or in the Parameters tab.
- **Run Model with the Specified PARVAL1 Values:** When this option is selected, the maximum number of optimization iterations (see NOPTMAX in the Control Data tab) will be set to 0. PEST will be used to run the model with the estimated parameter values, and it will terminate execution after just one model run. This setting can thus be used when you wish to calculate the objective function corresponding to a parameter set and/or to inspect observation residuals corresponding to that parameter set.
- **Regularization:** To run PEST in the regularization mode, you must supply at least one prior information equation with the name of the observation group (OBGNME) starting with “regul”. As described in (Doherty, 2018), *when run in “regularization” mode, PEST employs two objective functions rather than one. One of these is called the “measurement objective function” while the other is referred to as the “regularization objective function”. The lower is the former, the better do model-calculated quantities match their field-measured counterparts. The lower is the latter, the better do parameter values respect mathematical expressions of expert knowledge encapsulated in observations or prior information equations that are dedicated to this purpose.* Refer to the PEST book (Doherty, 2015) and Chapter 9 of

the Part I of the PEST Manual (Doherty, 2018) for details about the regularization mode. If the Pilot Points method is used, PM can instruct PPKREG1 of Groundwater Data Utilities (Doherty, 2020) to insert regularization information equations to the pest control data file prior to commencing a PEST run. See Section 3.4.2.4 for details.

- **Stochastic (PESTPP-IES):** When this option is selected, PM uses PESTPP-IES to generate parameter realizations that calibrate the model. The contents of the Run Management tab (see Section 3.9.5.2.8) and the following lines are added to the end of the PEST control file (pest.pst) that provide additional instructions to PESTPP-IES.
 - ++ies_num_reals(n): This line set the desired number of calibrated realizations to **n**, which is defined in *IES_NUM_REALS* of the Control Data tab.
 - ++ies_subset_size(m): This line instructs PESTPP-IES to devote **m** realizations to determining the best Marquardt lambda and line search factor to use during each iteration. The value of **m** is defined in *IES_SUBSET_SIZE* of the Control Data tab.
- 2. **Solver:** The selected solver is used to solve the linearized inverse problem during each iteration of the overall inverse process. As discussed in (Doherty, 2015), the Singular value decomposition (SVD) and the LSQR algorithm introduce numerical stability to solution of an ill-posed inverse problem. In most cases, *SVD Mode1*, *SVD Mode 2*, or *LSQR* should be used instead of the PEST's default solver. *SVD Mode 2* is superior to *SVD Mode 1* for cases involving many observations as the formation of the matrix for the inverse problem may take a considerable amount of time under these circumstances. A benefit of LSQR over SVD is its speed. If parameters number more than a couple of thousand, SVD can become very slow. This is not the case for LSQR which can happily handle tens of thousands of parameters and hundreds of thousands of observations.
- 3. **More Actions:** This menu acts on the Parameter Table (see below), the functions of the menu items are as follows.
 - **Update PARVAL1:** This action updates the initial values of zone-wise and pilot point parameters with a parameter value file, which is a text file created by PEST, but can also be constructed manually. You can run models based on parameter value files, see Section 3.9.5.4.5 for details.
 - **Apply PARVAL1:** This action updates the parameter values in model cells based on the PARVAL1, SCALE, and OFFSET values of zone-wise and pilot point parameters. The action copies the result of (PARVAL1 x SCALE + OFFSET) of zone-wise parameters to the model cells of respective zones. Likewise, the action uses the Kriging method to interpolate the values of (PARVAL1 x SCALE + OFFSET) of pilot points to model cells. If the *LAYERWEIGHTOPT* box in the Control Data tab is checked, the layer-weight values of multi-layer observations will be updated based on updated HK or T values. See next section for details about PARVAL1, SCALE and OFFSET.
 - **Clean Up:** Delete unused parameters from the Parameter Tab. An adjustable parameter becomes unused when its parameter number no longer exists in all cells of the respective parameter type.
 - **Activate Parameters:** Activate the parameters of the selected row(s) in the Parameter Table.

- **Deactivate Parameters:** Deactivate the parameters of the selected row(s) in the Parameter Table.
- **Remove Parameters:** Remove the selected row(s) from the Parameter Table.
- **Export Parameters**
- **Import Parameters**
- **Search for Parameters:** Search for adjustable parameters and add them to the Parameter Tab. Note that parameters defined by Pilot Points will appear in the Parameters Tab as grouped by model layers (see PARNME below for details).

3.9.5.2.1. Parameters Tab/Parameters Table

The Parameters tab contains a Parameters Table that lists the adjustable parameters and their properties. For parameters defined by pilot points, only the values of first three columns are used, all other values are defined for individual pilot points in the Pilot Point Parameters dialog box (see Figure 29 on page 44).

1. **PARNME** contains the predefined parameter names. Modification to PARNME is not allowed. The following naming convention of PARNME applies.
 - **For zone-wise parameters:** PARNME is a combination of the parameter number and the short name of the aquifer parameter (i.e., HK, HANI, VK, VANI, SS, SY, T, S or VC⁶⁷) or package (i.e., RCH, EVT, etc.). For example, if parameter numbers 1 and 2 are specified for the Recharge Package, the parameter names are RCH_1 and RCH_2. Similarly, if parameter numbers 3 is assigned to HK, the parameter name is HK_3. Note that when switching between groundwater flow packages (i.e., BCF, LPF, and UPW), some aquifer parameters might become not adjustable. For example, PEST cannot estimate values of T and S when using the LPF Package, because T and S are not included in the input file to the LPF Package.
 - **For pilot points:** Pilot points are grouped by layers. Each group of pilot points is represented by a row in the Parameters Table. For each group, PARNME is a combination of the short name of the aquifer parameter (i.e., HK, HANI, VK, VANI, SS, SY, T, S or VC⁶⁷) and the model layer. For example, HK1P represents all pilot points of HK in layer 1. PARNME is used as the prefix to the actual parameter names of individual pilot points. For example, HK1P12 is the parameter name of the pilot point #12 of HK in layer 1.
2. ☐: Check/uncheck the boxes in this column to activate/deactivate parameters. The value of a parameter will only be estimated if its box checked. Otherwise, the user-specified cell values will be used for the simulation. For parameters defined by pilot points, checking/unchecking a box in this column will activate/deactivate all pilot points of the respective aquifer parameters in the corresponding model layer.
3. **Description** contains optional text string to describe the parameter.

⁶⁷ Due to the length limitation of parameter names imposed by PEST, VC is used instead of VCONT. The default parameter names meet the PEST requirement that each parameter name must be unique and of twelve characters or less in length. PESTPP-IES and PESTPP-GLM allow for parameter names of up to 200 characters.

4. **PARVAL1**⁶⁸ is the initial value of the parameter. The default value is the arithmetical mean of the cell values of that parameter.
5. **PARLBND and PARUBND**⁶⁸: The parameter's lower bound (PARLBND) and upper bound (PARUBND) default to three orders of magnitude lower and higher than PARVAL1, respectively. For adjustable parameters, PARVAL1 must lie between these two bounds. For fixed and tied parameters, PARLBND and PARUBND are ignored.
6. **SD**⁶⁸: This is the standard deviation of the parameter. If PARTRANS (see below) is log-transformed, then SD pertain to the log of that parameter. This value is used only if the Operation Mode is *Stochastic (PESTPP-IES)*. The value will be cited in the parameter uncertainty file (pest.paramunc) that can be used by PESTPP-IES through the inclusion of the line *++parcov(pest.paramunc)* in the PEST control file (pest.pst).
7. **PARTRANS** controls the parameter transformation.
 - **None**: No transformation takes place.
 - **Log-transformed** ensures that the estimated value is always positive and is recommended for transmissivities and hydraulic conductivities. A parameter, which can become zero or negative during the parameter estimation process, must not be log-transformed; hence if a parameter's lower bound is zero or less, PEST will disallow logarithmic transformation for that parameter. The values of PARVAL1, PARUBND, PARLBND must pertain to the parameter itself (i.e., not log-transformed). At the end of the inversion process, PEST provides the optimized parameter value itself rather than the log of its value.
 - **Fixed**: The parameter is fixed at PARVAL1 and takes no part in the optimization process.
 - **Tied**: The parameter is linked to another parameter. A tied parameter plays only a limited role in the estimation process. The *parent* parameter, to which the tied parameter is linked, takes an active part in the parameter estimation process and must be neither fixed nor tied itself. During the parameter estimation process, the value of the tied parameter maintains the same ratio to the parent parameter as the ratio of their initial values.
8. **PARCHGLIM** is used to designate whether the value change of an adjustable parameter is relative-limited or factor-limited. See the discussion on RELPARMAX and FACPARMAX in the Control Data Tab below. PARCHGLIM is not used for a fixed parameter.
9. **PARGP** is the number of the parameter group to which a parameter belongs. Parameter groups are discussed below.
10. **PARTIED** is the parent parameter to which is parameter is linked.
11. **SCALE** and **OFFSET**: Just before a parameter value is written to an input file of MODFLOW, it is multiplied by SCALE, after which OFFSET is added. The use of these two variables allows you to redefine the domain of a parameter. Because they operate on the

⁶⁸ For log-transformed parameters, the values of PARVAL1, PARUBND, PARLBND, and SD must be greater than zero and must pertain to the parameter itself (not log-transformed).

parameter value “at the last moment” before it is sent, they take no part in the estimation process. Using SCALE and OFFSET, the parameter b_p optimized by PEST is

$$b_p = (b_m - OFFSET)/SCALE \quad (56)$$

where b_m is the parameter seen by the model. If you wish to leave a parameter unaffected by SCALE and OFFSET, enter the SCALE as 1.0 and the OFFSET as 0.0.

3.9.5.2.2. Parameter Group Tab

In PEST, the input variables that define how derivatives are calculated pertain to parameter groups rather than to individual parameters. Figure 147 shows the Parameter Group table that is used to enter those input variables. The columns of the Parameter Group table are as follows.

1. **Description** is an optional description of the parameter group.
2. **INCTYP and DERINC:** INCTYP is the method by which parameter increments are calculated. DERINC is the absolute or relative parameter increment. The option for INCTYP are as follows:
 - **Relative:** The parameter increment is calculated as a fraction of the current value of that parameter; that fraction is specified in DERINC. A DERINC value of 0.01 is often appropriate.
 - **Absolute:** The parameter increment is fixed at the value of DERINC. No suggestion for an appropriate DERINC value can be provided for this option; the most appropriate increment will depend on the parameter magnitudes.

Parameters	Description	INCTYP	DERINC	DERINCLB	FORCEN	DERINCMUL	DERMTHD	SPLITTHRESH	SPLITRELDIFF	SPLITACTION
1	HK	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
2	VK	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
3	SS	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
4	SY	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
5	RCH	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
6	HFB	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
7	FHB FLOW	relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
8	FHB HEAD	absolute	2	0	switch	1	parabolic	0.0001	0.5	smaller
9	SFR_HC	relative	0.1	0	switch	1	parabolic	0.0001	0.5	smaller
10		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
11		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
12		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
13		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
14		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
15		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
16		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller
17		relative	0.01	0	switch	1	parabolic	0.0001	0.5	smaller

Figure 147. The Parameter Group tab of the PEST Settings dialog box

- **Rel_to_max:** The parameter increment is calculated as a fraction of the group member with highest absolute value, that fraction again being DERINC. A DERINC value of 0.01 is often appropriate. If a group contains members, which are fixed and/or tied, the values of these parameters are used when calculating parameter increments.
3. **DERINCLB** is the lower bound of relative parameter increment; ignored if INCTYP is *Absolute*. If a parameter increment is calculated as *Relative*, it may become too low if the parameter values become very small. And, if a parameter increment is calculated as *Rel_to_max*, it may become too low if the modulus of the largest parameter in the group is very small. A parameter increment becomes “too low” if it does not allow reliable derivatives to be calculated with respect to that parameter because of round off errors incurred in the subtraction of nearly equal model generated values. DERINCLB is used to bypass this possibility. Set DERINCLB to zero if you do not wish to place a lower limit on parameter increments in this fashion.
 4. **FORCEN** determines whether higher order derivatives calculation is undertaken.
 - **Always_2:** Derivatives will be calculated using the forward difference method.
 - **Always_3:** PEST will use the central (i.e., three-point) method to calculate the derivatives. In this case, twice as many model runs as there are parameters within the group will be required. However, the derivatives will be calculated with greater accuracy and this will probably have a beneficial effect on the performance of PEST.
 - **Always_5:** PEST will use a five-point method to calculate the derivatives.
 - **Switch:** Derivatives calculations will begin using the forward difference method, switching to the central (i.e., three-point) method for the remainder of the estimation process after the relative objective function reduction between successive iterations is less than PHIREDSWH as defined in the Control Data Tab below.
 - **Switch_5:** The switch is made from forward differences to five-point derivatives under the same conditions as described in Switch above.
 5. **DERINCMUL** is the parameter increment multiplier; this is the value by which DERINC is multiplied when it is used to evaluate increments for any of the three central derivatives methods. Sometimes it is useful to employ larger increments for central derivatives calculation than for forward derivatives calculation. However, if the increment is raised too high, derivative precision must ultimately fall. Note that through DERINCMUL you can also reduce the increment used for central derivatives calculation if you wish. DERINCMUL plays a similar role in implementation of five-point derivatives as that which it plays in implementation of central derivatives.
 6. **DERMTHD** defines the method of higher order derivatives calculation.
 - If *FORCEN* is set to *always_3* or *switch*, *DERMTHD* must be set to one of *outside_pts*, *parabolic* or *best_fit*; this determines the method of central derivatives calculation to be used by PEST.
 - If *FORCEN* is set to *aways_5* or *switch_5*, *DERMTHD* must be either *minvar* to activate the minimum error variance alternative of the five-point methodology or to *maxprec*

to activate the maximum precision alternative. Refer to Part I of the PEST Manual (Doherty, 2016) for details about these methods.

7. **SPLITTHRESH, SPLITRELDIFF, and SPLITACTION:** These variables are used by “split slope analysis”. See Section 3.5.7 of the Part I of the PEST Manual (Doherty, 2016) for details.

3.9.5.2.3. Prior Information Tab

The Prior Information tab⁶⁹ (Figure 148) contains information concerning the parameters that you wish to optimize based on existing knowledge, previous studies, expert opinions, or other sources of information that might inform the likely values of the parameters. This information may be in the form of other unrelated estimates of some or all parameters, or of relationships between parameters. It is often useful to include this information in the parameter estimation process because it may lend stability to the process.

To define prior information, first check the Active box in the Prior Information tab, and then enter the prior information equation in the Prior Information column. The syntax of a prior information line is

$$PILBL PIFAC * PARNME + PIFAC * \log(PARNME) \dots = PIVAL WEIGHT OBGNME \quad (57)$$

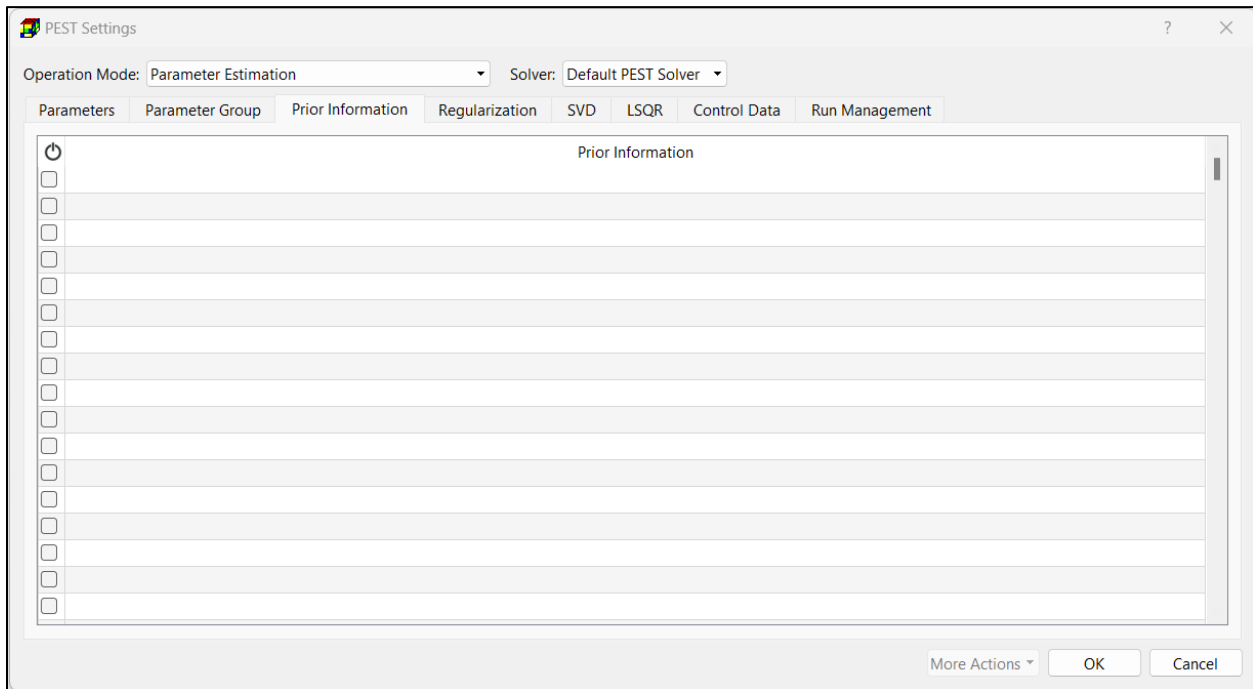


Figure 148. The Prior Information tab of the PEST Settings dialog box

⁶⁹ If the Pilot Points method is used, PM can instruct PPKREG1 of Groundwater Data Utilities (Doherty, 2020) to insert regularization prior information equations to the pest control data file prior to commencing a PEST run. See Section 3.4.2.4 for details.

The variables of the prior information equations are defined as follows. All variables and symbols must be separated from by at least one space.

- **PILBL:** Each prior information article must begin with a case insensitive prior information label. The label must be no more than twenty characters in length and must be unique to each prior information article.
- **PIFAC and PARNME:** *PIFAC* is a parameter factor. *PARNME* is the parameter name. Both are required. To the left of the “=” sign there are one or more combinations *PIFAC* and *PARNME*, with a “log” prefix to *PARNME* if appropriate. *PIFAC* and *PARNME* are separated by a “*” character signifying multiplication. All parameters referenced in a prior information equation must be adjustable parameters, i.e., you must not include any fixed or tied parameters. Furthermore, any parameter can be referenced only once in any one prior information equation; however, it can be referenced in more than one equation.
- **PIVAL** is the value of the right-hand side of the prior information equation.
- **WEIGHT:** The prior information weight should ideally be inversely proportional to the standard deviation of *PIVAL*; it can be zero if you wish but must not be negative. In practice the weights should be chosen such that the prior information equation neither dominates the objective function or is dwarfed by other components of the objective function. In choosing observation and prior information weights, remember that the weight is multiplied by its respective residual and then squared before being assimilated into the objective function.
- **OBSGNME** is observation group to which the prior information belongs. *OBSGNME* must be twelve characters or less in length. When running PEST in the Regularization mode, *OBSGNME* of at least one of the prior information equations must start with “regul”. PEST can accommodate multiple regularization groups. Any observation group name starting with “regul” is a regularization group.

Some examples of prior information equations are given below, refer to the Part I of the PEST Manual (Doherty, 2016) for more details.

Example 1:

```
pi1 1.0 * par1 = 2.305 1.0 regulg1
```

This prior information equation is given the label *pi1* and has a single untransformed parameter named *par1*. This parameter has a preferred value of 2.305. This information will be included in the optimization process with a weight of 1.0 and is assigned to the observation group *regulg1*.

Example 2:

```
pi1 1.0 * log(par1) = 0.362671 1.0 prgp
```

This prior information equation is similar to the previous example, with the exception that *par1* is log-transformed. The preferred value of the log-transformed parameter is 0.362671 (= log(2.305)). Note that logs are taken to base 10.

Example 3:

pi2 1.0 * par + 3.4 * par3 - 2.3 * par4 = 1.09e3 3.0 prgp

This example shows that the left side of a prior information equation can be comprised of the sum and/or difference of a number of factor-parameter pairs; these pairs must be separated from each other by a “+” or “-” sign, with a space to either side of the sign.

Example 4:

pi3 1.0 * log(par1) - 1.0 * log(par2) = 1.60206 2.0 prgp

Prior information equations which include log-transformed parameters must express a relationship between the logs of those parameters. This equation defines the ratio between the estimated values of parameters “par1” and “par2” to be about 40.0, i.e., $\log(\text{par1}/\text{par2}) = \log(40) = 1.60206$.

3.9.5.2.4. Regularization Tab

In its broadest sense, regularization is a term used to describe the process whereby many parameters can be simultaneously estimated without incurring the numerical instability that normally accompanies parameter nonuniqueness. Numerical stability is normally achieved through the provision of supplementary information to the parameter estimation process. Such supplementary information often takes the form of preferred values for parameters, or for relationships between parameters (i.e., prior information).

Regularization is particularly useful in estimating values for parameters which describe the spatial distribution of some property over a two- or three-dimensional model domain of a ground water model. You are no longer required to subdivide the model domain into a small number of zones of piecewise parameter constancy. Rather, a large number of parameters can be used to describe

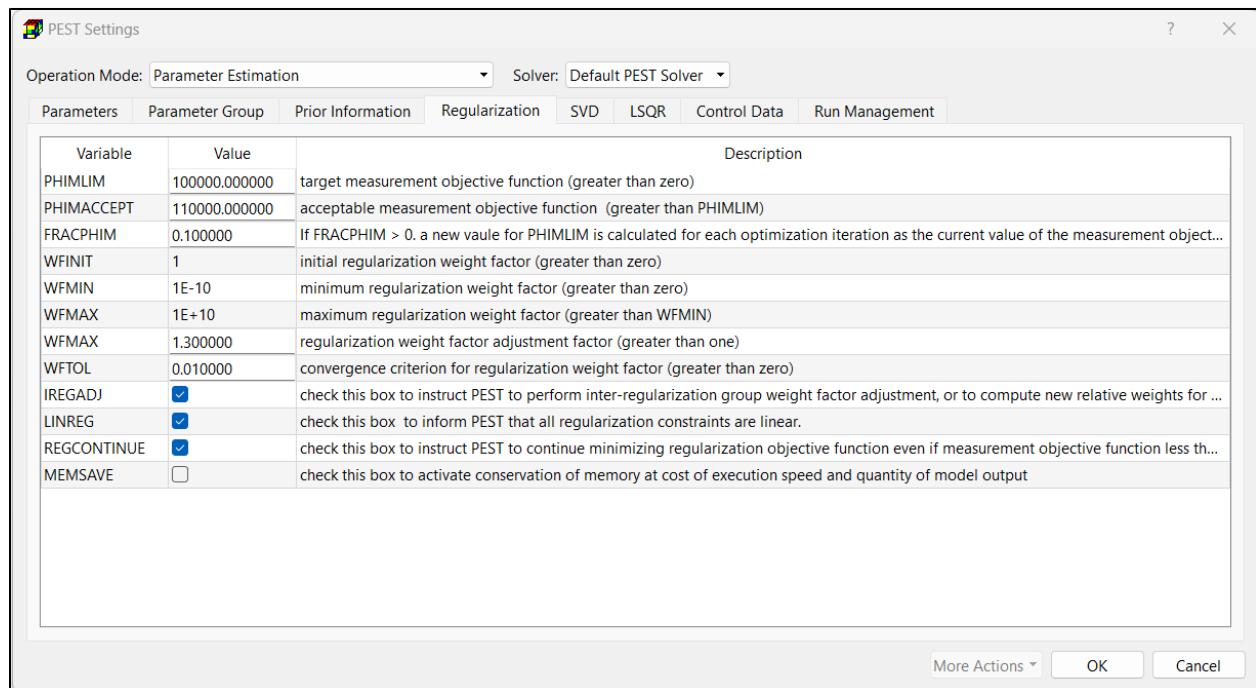


Figure 149. The Regularization tab of the PEST Settings dialog box

the distribution of the spatial property and PEST's regularization functionality can be used to estimate values for these parameters.

To run PEST in the regularization mode⁷⁰, select Regularization from the Operation Mode dropdown box, define at least one prior information equation and set the observation group (OBGNME) of the prior information equation to start with "regul", then assign appropriate values to the following parameters in the Regularization Tab (Figure 149).

1. **PHIMLIM** is the measurement objective function (i.e., sum of squared weighted residuals) that PEST aims for while keeping the regularization objective function as low as possible. PHIMLIM must be greater than zero.
2. **PHIMACCEPT** is the acceptable measurement objective function and should be set 5-10% greater than PHIMLIM. During each optimization iteration, PEST calculates the optimal value of the regularization weight factor for that iteration. That optimal value would result in a measurement component objective function equal to PHIMLIM. However, due to the approximate nature of the linearity assumption, PEST may not be able to lower the measurement objective function to PHIMLIM on that iteration. If it cannot lower the measurement objective function to PHIMLIM, it simply proceeds to the next optimization iteration and tries again. However, if it does succeed in lowering the measurement objective function to the acceptable level, then PEST will attempt to lower the regularization objective function while maintaining the measurement objective function below PHIMACCEPT.
3. **FRACPHIM** allows you to set PHIMLIM very low (possibly lower than is achievable), but still retain the benefits of regularization. FRACPHIM must be greater or equal to 0 and less than 1. The recommended value for FRACPHIM is 0.1. If FRACPHIM is zero, then it has no effect on the inversion process. If FRACPHIM lies between 0.0 and 1.0, then PEST calculates a new value for PHIMLIM at the beginning of each optimization iteration. That value is calculated as the current value of the measurement objective function times FRACPHIM, or the user-supplied value of PHIMLIM, whichever is higher. Thus, PEST will always aim for a measurement objective function that is lower than the current one. However, it does not pursue a target that is so low as to require the complete abandonment of regularization.
4. **WFINIT** is the initial regularization weight factor and must be greater than zero. During every optimization iteration PEST calculates a suitable regularization weight factor using an iterative, numerical solution procedure. If there are many adjustable parameters, calculation of the regularization weight factor for the first iteration can be time-consuming if WFINIT is far from optimal. Hence if you have any idea of what the weight factor should be (for example from a previous PEST run), then you should provide WFINIT with this value. Otherwise simply set it to 1.0.

⁷⁰ If Pilot Points are used, PM can insert prior information and regularization data to the PEST control file by checking the *Include RegInfo* box in Figure 31. See

5. **WFMIN** and **WFMAX** are respectively the minimum and maximum regularization weight factors and must be greater than zero. If a regularization scheme is poor, (and does not lend too much stability to an already unstable parameter estimation process), selection of appropriate values for WFMIN and WFMAX may be quite important, for these can prevent PEST from calculating outrageous values for the regularization weight factor to compensate for inadequacies of the regularization scheme.
6. **WFFAC** and **WFTOL**. WFFAC is the regularization weight factor adjustment factor and must be greater than one. WFTOL is the convergence criterion for regularization weight factor and must be greater than zero. When PEST calculates the appropriate regularization weight factor to use during an optimization iteration, it uses an iterative procedure which begins at the value of the regularization weight factor calculated for the previous optimization iteration; for the first iteration, it uses WFINIT to start the procedure. In the process of finding the weight factor, PEST first travels along a path of progressively increasing or decreasing weight factor by either multiplying or dividing the weight factor by WFFAC; it continues to do this until it has found two successive weight factors which lie on either side of the optimal weight factor for that optimization iteration. Once it has done this, it uses Newton's method to calculate the optimal weight factor, through a series of successive approximations. When two subsequent weight factors calculated in this way differ from each other by no more than a relative amount of WFTOL, the optimal weight factor is deemed to have been calculated.
7. **IREGADJ**: Check this box to instruct PEST to perform inter-regularization group weight factor adjustment, or to compute new relative weights for regularization observations and prior information equations
8. **LINREG**: Check this box to inform PEST that all regularization constraints are linear. As discussed in the PEST Manual, regularization constraints can be supplied through observations, through prior information, or both. Prior information relationships are always linear. Regularization constraints supplied as observations (for which the current value of pertinent relationships is calculated by the model), can be linear or nonlinear; in either case, derivatives of these relationships with respect to adjustable parameters are re-evaluated by PEST during each optimization iteration. If regularization information is entirely linear, many matrix operations carried out as part of PEST's regularization functionality do not need to be repeated from iteration to iteration. If repetition of these calculations can be avoided in parameter estimation contexts involving many regularization constraints, significant gains in efficiency can be made.
9. **REGCONTINUE**: Check this box to instruct PEST to continue minimizing regularization objective function even if measurement objective function less than PHIMLIM. Under normal circumstances, when working in regularization mode, PEST ceases execution immediately if the measurement objective function falls below PHIMLIM. There are some circumstances, however, where minimization of the regularization objective function is just as important as allowing the measurement objective function to reach PHIMLIM. If this box is checked, PEST will continue optimizing regularization objective function after reached PHIMLIM.

10. MEMSAVE: Check this box to activate conservation of memory at cost of execution speed and quantity of model output. If this box is checked, nonessential PEST tasks which are curtailed include the following:

- The parameter covariance matrix, and matrices derived from it, are not calculated by PEST at regular intervals during the parameter estimation process for recording in the matrix file, pest.mtt; nor are these matrices calculated at the end of the inversion process for recording in the run record file pest.rec. Because the covariance matrix is unavailable, parameter uncertainties cannot be calculated, and hence are also not recorded in the run record file. (In a regularization context, these have little meaning anyway.)
- Some avenues for increasing the efficiency of regularization calculations are no longer available under the leaner storage regime that prevails when memory conservation is active (including the benefits gained through the LINREG variable and through the placing of regularization observations behind other observations involved in the parameter estimation process). This can lead to significant run-time penalties in problems involving many parameters; unfortunately, these are the very contexts in which memory conservation is most likely to be warranted.

3.9.5.2.5. SVD Tab

The variables of the SVD tab (Figure 150) are used only if the Solver is either *SVD Mode 1* or *SVD Mode 2*.

1. **MAXSING** is the number of singular values at which truncation occurs and must be greater than zero. Ideally this should be equal to the optimal dimensionality of the solution space. This is difficult to determine though the SUPCALC utility described in part II of the PEST Manual (Doherty, 2016) may provide some help in this regard. Normally the role of singular value decomposition is to achieve numerical stability of the inversion process rather than to provide regularization. Hence, the EIGTHRESH (see below) variable should be used to set the point of singular value truncation. To achieve this, set MAXSING to a number equal to, or greater than, the maximum number of adjustable parameters. This is the best setting to use in most cases.
2. **EIGTHRESH** is the eigenvalue ratio threshold for truncation ($0 \leq \text{EIGTHRESH} < 1$). To ensure numerical stability of the inversion process this should be set to $5E-7$ or above, for this is the value at which numerical noise can be amplified to a level where it smothers the “signal” contained in the calibration dataset. See Section 4.4 of the Part I of the PEST Manual (Doherty, 2016) for details.
3. **EIGWRITE:** Check this option to save singular values and eigenvectors to the SVD output file pest.svd. If unchecked, only singular values are saved. The SVD output file can become very large and not all the information recorded in this file is always worth reading. However sometimes an inspection of singular values can assist in understanding the degree to which an inverse problem is ill-posed.

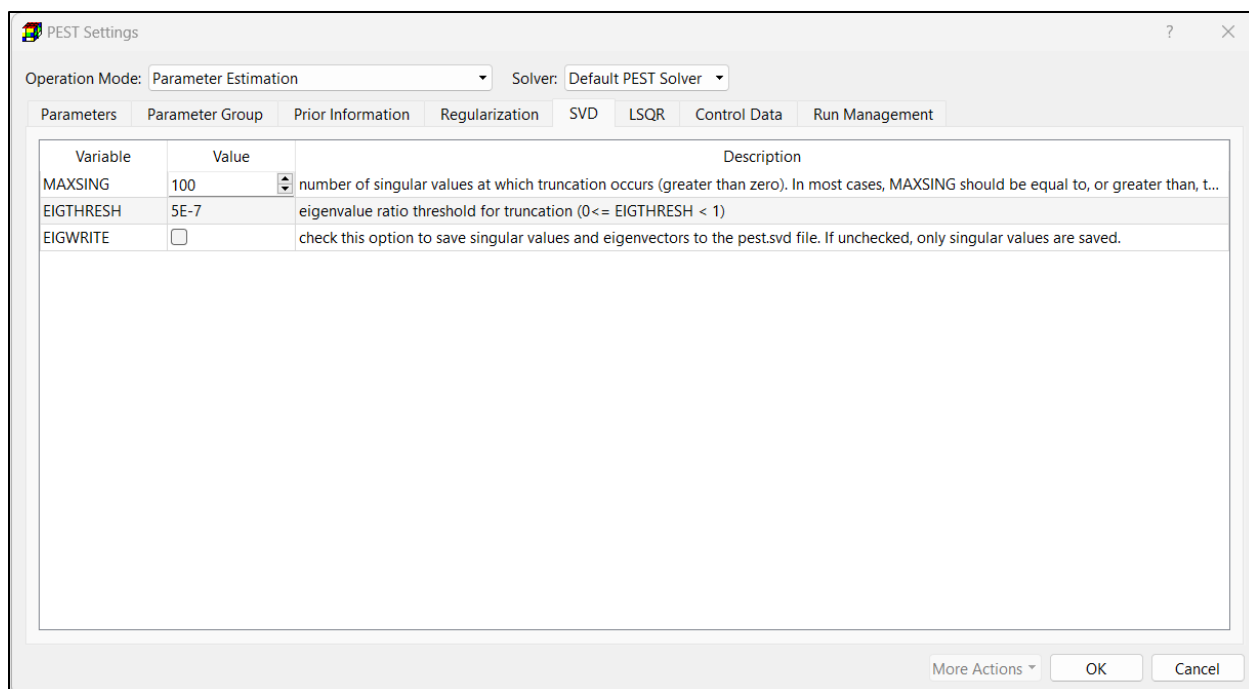


Figure 150. The SVD tab of the PEST Settings dialog box

3.9.5.2.6. LSQR Tab

The variables of the LSQR tab (Figure 151) are used only if the Solver is LSQR.

1. **LSQR_ATOL:** This is the LSQR atol input variable and must be zero or greater. This variable is defined as “An estimate of the relative error in the data defining the matrix **A**. For example, if **A** is accurate to about 6 digits, set atol to 1.0e-6.”
2. **LSQR_BTOL:** This is the LSQR btol input variable and must be zero or greater. This variable is defined as “An estimate of the relative error in the data defining the rhs vector **b**. For example, if **b** is accurate to about 6 digits, set btol to 1.0e-6.”
3. **LSQR_CONLIM:** This is the LSQR *conlim* input variable and must be zero or greater. A value of 1000.0 generally works well with PEST. See Section 4.5.2 of the Part I of the PEST Manual (Doherty, 2016) for the definition of *conlim*.
4. **LSQR_ITNLIM:** This is the LSQR *itnlim* input variable and must be greater than 0. This value should be set to four times the number of adjustable parameters as recommended by (Doherty, 2016).
5. **LSQR_WRITE:** Check this box to instruct PEST to write output from the LSQR solver to the pest.lsq file. Information from each LSQR call is appended to this file; hence it can become quite lengthy.

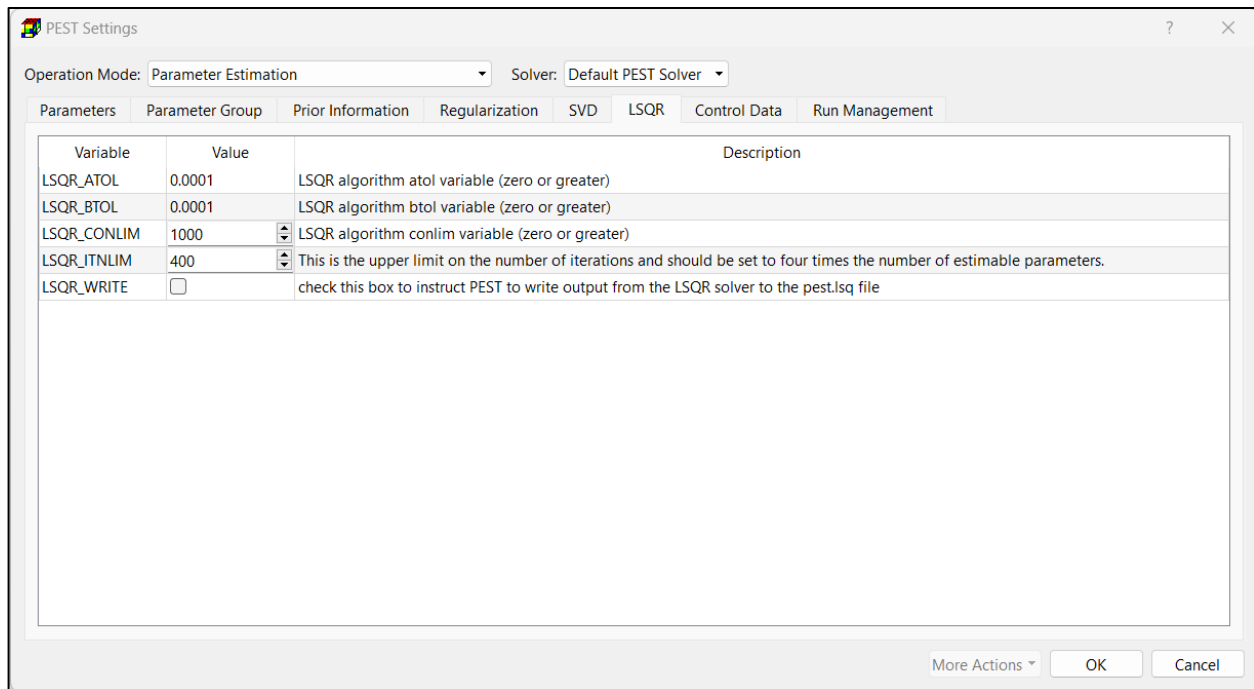


Figure 151. The LSQR tab of the PEST Settings dialog box

3.9.5.2.7. Control Data Tab

The Control Data tab (Figure 152) contains entries that are used in the Control Data section of a PEST control file. This section provides a brief explanation of the entries. See Section 4.2 of the Part I of the PEST Manual (Doherty, 2016; 2018) for details.

1. **PRECIS** defines the format for writing parameter values to model input files.
2. **DPOINT**: Select *nopoint* to omit decimal point from representations of parameter values on model input files. Select *point* to ensure that the decimal point is always present.
3. **RLAMBDA1** is the value of initial Marquardt lambda and must be zero or greater. A value of 10.0 is appropriate in most cases. However, if PEST complains about the normal matrix being singular you may need to raise it.
4. **RLAMFAC** is the factor by which PEST adjusts the Marquardt lambda. RLAMFAC must be greater than 1.0; a value of 2.0 seems to work well on many occasions. When PEST reduces lambda, it divides by RLAMFAC; when it increases lambda, it multiplies by RLAMFAC.
5. **PHIRATSUF** is the fractional objective function sufficient for end of current iteration. PHIRATSUF must be between 0 and 1. A value of 0.3 is appropriate in most cases. During an iteration of the inversion process, PEST may calculate a parameter upgrade vector using several different Marquardt lambdas. First it lowers lambda and, if this is unsuccessful in lowering the objective function, it then raises lambda. If, at any stage, it calculates an objective function which is a fraction PHIRATSUF or less of the starting objective function for that iteration, PEST considers that the goal of the current iteration has been achieved and moves on to the next iteration.

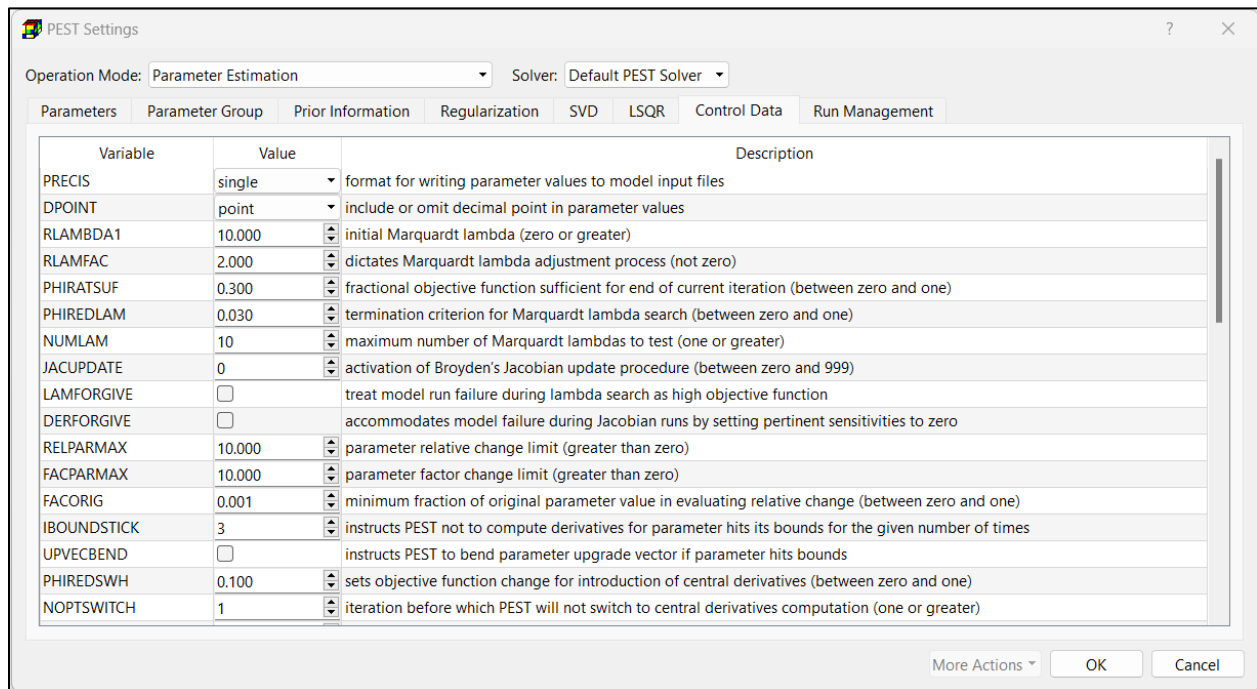


Figure 152. The Control Data tab of the PEST Settings dialog box

6. **PHIREDLAM** is the termination criterion for Marquardt lambda search. PHIREDLAM must be between 0 and 1. If the relative reduction in the objective function between the use of two consecutive lambdas is less than PHIREDLAM, PEST moves to the next iteration.
7. **NUMLAM** is the maximum number of Marquardt lambdas to test during an iteration. NUMLAM must be 1 or greater and should normally be set to 10.
8. **JACUPDATE** is used in conjunction with the activation of Broyden's Jacobian update procedure, which provides a mechanism for improving the Jacobian matrix based on model outputs calculated during model runs undertaken for testing parameter upgrades calculated using different values of the Marquardt lambda.
 - If it is set to zero, Broyden updating of the Jacobian matrix does not occur.
 - If it is set to 999, Jacobian matrix is updated after the first parameter upgrade attempt, and every parameter upgrade attempt thereafter.
 - If it is set to n , where $0 < n < 999$, Broyden updating will be undertaken during a PEST iteration following the first n attempts to upgrade parameters using n different values of the Marquardt lambdas.
9. **LAMFORGIVE** treats model run failure during lambda search as high objective function.
 - If this box is unchecked: when a model fails to run to completion during a parameter estimation process, PEST reports this error condition and ceases execution with an error message.
 - If this box is checked: when a model fails to run to completion during a parameter estimation process, PEST ignores the failed model run and attempts new sets of parameters using other values of the Marquardt lambda. If it failed to find parameters

that allow the model to complete execution, PEST either switches to central derivatives calculation or ceases execution with an appropriate error message.

- 10. DERFORGIVE** plays a similar role to LAMFORGIVE. However, DERFORGIVE governs PEST's behavior when model runs are being undertaken for finite-difference derivatives calculation. If this box is checked, DERFORGIVE accommodates model failure during Jacobian runs by setting pertinent sensitivities to zero.
- 11. RELPARMAX** is the maximum relative change that a parameter may undergo between optimization iterations. RELPARMAX must be greater than zero.
- 12. FACPARMAX** is the maximum factor change that a parameter may undergo between optimization iterations. FACPARMAX must be greater than zero.
- 13. FACORIG:** If the absolute value of a parameter falls below FACORIG times its original value, then FACORIG times its original value is used in evaluating relative change. FACORIG is not used to adjust limits for log-transformed parameters. FACORIG must be greater than zero. A value of 0.001 is often suitable.
- 14. IBOUNDSTICK** instructs PEST not to compute derivatives for parameter hits its bounds for the given number of times.
- 15. UPVECBEND:** If this box is checked, PEST will bend the parameter upgrade vector if a parameter hits its bounds. Normally, this box should be unchecked.
- 16. PHIREDSWH:** If FORCEN (see Parameter Group Tab above) is set to "switch" or "switch_5", and if the relative reduction in the objective function between successive iterations is less than the PHIREDSWH, PEST will switch from single to three-point or five-point derivatives calculation. A value of 0.1 is often suitable for PHIREDSWH.
- 17. NOPTSWITCH** is the iteration before which PEST will not switch to higher order derivatives computation. NOPTSWITCH must be one or greater. A value of 3 or 4 often works well. If in doubt, set the value to 1.
- 18. SPLITSWH** determines when, in the inversion process, split slope analysis begins. split slope analysis will begin when the ratio of the best objective function achieved during a certain iteration to that achieved in the previous iteration is equal to or higher than SPLITSWH. If SPLITSWH is provided with a zero or negative value it is ignored. Normally, SPLITSWH should be set to zero, even if split slope analysis is activated.
- 19. DOAUI:** Check this box to instruct PEST to implement automatic user intervention. Only on very rare occasions is automatic user intervention useful. Insensitivity of one or a number of parameters is accommodated far more gracefully using singular value decomposition or LSQR as a solution device for the inverse problem.
- 20. DOSENREUSE:** Check this box to instruct PEST to reuse parameter sensitivities. On most occasions, there is little benefit to be gained from invoking PEST's sensitivity reuse functionality. For details, see Section 7 of the Part I of the PEST Manual.
- 21. BOUNDSSCALE:** If this box is checked, parameters are scaled by the inter-bounds interval if using SVD or LSQR as the solver. Experience has shown that on most occasions scaling by bounds is worth a try.

- 22. NOPTMAX** is the maximum number of optimization iterations that allows PEST to find a solution that adequately minimizes the difference between observed and simulated values. PEST ceases execution after NOPTMAX iterations have elapsed. NOPTMAX should normally be greater than zero. NOPTMAX should be less than or equal to 3 for PESTPP-IES, as recommended by its internal sanity-check. The best value for NOPTMAX can vary depending on the specific problem and model you are working with. In addition, the following special values can be used.
- If NOPTMAX is set to 0, PEST will not estimate parameters, nor even calculate a Jacobian matrix. Instead, it will terminate execution after just one model run. This setting can be used to ensure that PEST setup is correct; at the same time, it can be used to compute objective function components and residuals based on initial parameter values.
 - If NOPTMAX is set to -2 PEST will calculate the Jacobian matrix, store it in the pest.jco file, and then cease execution immediately. This matrix can then be used for linear analysis, and/or for construction of an SVD-assist input dataset.
 - Setting NOPTMAX to -1 instructs PEST to compute the Jacobian matrix. However, after doing this, PEST records the same information on its output files as that which it would normally record on completion of an inversion process. This includes composite sensitivities and, if pertinent, post-calibration uncertainty and covariance statistics calculated from parameter sensitivities. It then undertakes a final model run so that model input and output files remaining after PEST execution is complete pertain to parameters values provided in the PEST control file.
- 23. PHIRESTP** and **NPHISTP**⁷¹: PHIRESTP is the relative objective function reduction triggering termination and must be greater than zero. NPHISTP is the number of successive iterations over which PHIRESTP applies. For many cases 0.005 and 4 are suitable values for PHIRESTP and NPHISTP respectively.
- 24. NPHINORED**⁷¹: If PEST has failed to lower the objective function over NPHINORED successive iterations, it will terminate execution. A value of 3 or 4 is often suitable.
- 25. RELPARSTP** and **NRELPAR**: If the magnitude of the maximum relative parameter change between iterations is less than RELPARSTP over NRELPAR successive iterations, PEST will cease execution. A value of 0.005 is often suitable for RELPARSTP. A value of 3 or 4 is normally satisfactory for NRELPAR.
- 26. PHISTOPTHRESH**: If PHISTOPTHRESH is set to a positive number, PEST will cease execution if the objective function falls below this value at the end of any iteration. If PHISTOPTHRESH is set to zero or a negative number, then it is ignored.
- 27. LASTRUN**: Check this box to instruct PEST to undertake final model run with best parameters.
- 28. PHIABANDON** is the objective function value at which to abandon optimization process. Set this to a very high number or to a non-positive number to deactivate this function.

⁷¹ PESTPP-IES applies the PHIRESTP, NPHISTP and NPHINORED termination criteria to the mean objective function calculated using all realizations of the ensemble over successive smoother iterations.

- 29. ICOV, ICOR, and IEIG:** If PEST is run in “estimation” mode, and if the Default PEST Solver is used in solution of the inverse problem, then at the end of each iteration of the inversion process PEST can write linear estimates of the posterior covariance and correlation coefficient matrices, as well as eigenvectors and eigenvalues of the posterior covariance matrix to the pest.mtt file. Check the boxes of ICOV, ICOR and IEIG to enable the output of the respective matrices.
- 30. IRES:** Check this box to instruct PEST to record resolution data to the pest.rsd file. This is a binary file which is used by utilities such as RESPROC, RESWRIT and PARAMERR which compute the resolution matrix, and quantify post-calibration predictive error variance.
- 31. JCOSAVE:** Check this box to instruct PEST to save best Jacobian file as the pest.jco file.
- 32. VERBOSEREC:** Check this box to instruct PEST to include parameter and observation data lists in the run record file. Uncheck this box so that PEST will write a much shorter run record file.
- 33. JCOSAVEITN:** Check this box to instruct PEST to write current Jacobian matrix to iteration-specific pest.jco.n file at the end of every optimization iteration, where n is the number of iteration.
- 34. REISAVEITN:** Check this box to instruct PEST to store best-fit residuals to iteration-specific residuals files pest.rei.n at end of every optimization iteration, where n is the iteration number.
- 35. PARSAVEITN:** Check this box to instruct PEST to store iteration specific parameter value files pest.par.n at end of every optimization iteration, where n is the iteration number.
- 36. PARSAVEITN:** Check this box to instruct PEST to creates a series of run specific parameter value files named pest.par.n_m where n is the run packet index and m is the run number within that packet. For identification purposes, run packet indices are also recorded on the run management record file pest.rmr. This file also records run numbers and the nodes to which runs were allocated. Hence a user can associate parameter values with the runs performed by different nodes involved in PEST parallelization.

The following entries are used by PM or PESTPP-IES and are not part of the original PEST control data section.

- 37. HEADOPT:** This option affects how the hydraulic head data are used for calibration with PEST.
- If this box is checked, the calculated head values are temporally interpolated to the observation times, and the interpolated head values are compared with the observed values during the calibration.
 - If this box is not checked, the observed head values are temporally interpolated to times at which the head values are calculated and saved. The interpolated observation values are compared with the calculated values during the calibration.
- 38. DDOWNOPT:** This option affects how the drawdown data are used for calibration with PEST. This option works the same way as HEADOPT.

- 39. STREAMFLOWOPT:** The streamflow data calculated by the SFR2 Package and saved by the Gage Package can be used for model calibration with PEST. This option affects how the streamflow data are used for calibration and works the same way as HEADOPT.
- 40. LAKESTAGEOPT:** The lake stage data calculated by the LAK3 Package and saved by the Gage Package can be used for model calibration with PEST. This option affects how the stage data are used for calibration and works the same way as HEADOPT.
- 41. LAYERWEIGHTOPT:** If this box is checked, the layer weight values for multi-layer head/drawdown observations⁷² are recalculated based on updated HK values during a PEST iteration. The updated layer weight value w' for a specific layer is calculated as $w' = w / HK * HK'$, where w is the original layer weight value, HK is the initial horizontal hydraulic conductivity, and HK' is the updated horizontal hydraulic conductivity during a PEST iteration. The updated layer weight values for all layers are normalized so that their sum equals 1. If this box is unchecked, the layer weight values will not be updated during a PEST run.
- 42. IES_NUM_REALS:** This is the desired number of parameter realizations that should be generated by PESTPP-IES.
- 43. IES_SUBSET_SIZE:** This is the number of realizations used by PESTPP-IES in testing and evaluation of different Marquardt lambdas. If supplied as a negative value, then the absolute value is treated as a percentage of the current ensemble size – this allows the subset size to fluctuate with the size of the ensemble. See the PESTPP-IES users guide⁷³ for details about the role of the Marquardt Lambda.

3.9.5.2.8. Run Management Tab

The Run Management tab (Figure 153) contains a text editor where you can enter control variables⁷³ for PESTPP-IES and PESTPP-GLM runs. If the *Operation Mode* is “Stochastic (PESTPP-IES)” or if the *PEST Version* is “PESTPP-GLM”⁷⁴, the contents of the text editor will be appended to the end of the PEST control file. The following lines are added to the text editor by default. Refer to the PESTPP Users Manual⁷³ for details.

- ++parcov(pest.paramunc):** this line instructs PESTPP-IES to use the parameter uncertainty file `pest.paramunc`. A parameter uncertainty file (Doherty, 2018) contains the standard deviation values of individual zonewise parameters and the covariance matrices of individual pilot points datasets. Standard deviation values are specified by the user in the Parameters tab. Covariance matrix files are created by using the PPCOV utility (Doherty, 2018). Prior to running PESTPP-IES, PM will create and run the Windows batch file `RunPPCOV.bat` that contains commands to run PPCOV for individual pilot points and geostatistical structure datasets. PM will also generate the `pest.paramunc` file and save it in the model data folder. The `RunPPCOV.bat` file and all associated input data files are saved in the *ppcov* and *kriging* subfolders of the model data folder.

⁷² See Section 3.6.3 for details of the layer weight values for multi-layer observations.

⁷³ https://github.com/usgs/pestpp/blob/master/documentation/pestpp_users_manual.md

⁷⁴ PEST Version is defined in the Misc. tab of the Preferences dialog box.

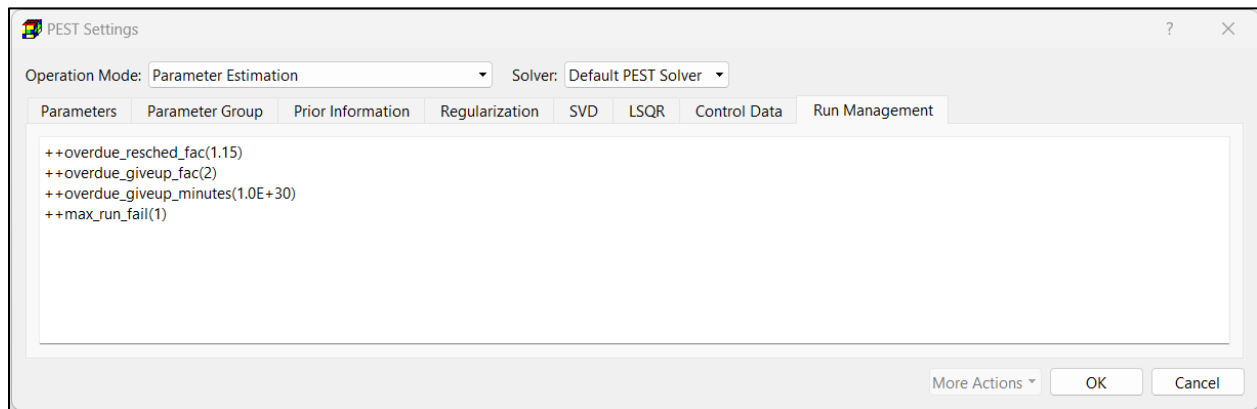


Figure 153. The Run Management tab of the PEST Settings dialog box

2. **++ies_save_binary(true)**: This line instructs PESTPP-IES to record iteration-specific parameter and observation ensembles in binary (i.e., JCB) format instead of CSV format.
3. **++overdue_resched_fac(1.15)**: If the run manager is still awaiting completion of a simulation that has already been running for more than X times the average model run time (calculated on the basis of previous successful runs), then it will ask another agent to start the same run if one is available. The maximum number of concurrent runs that employ the same set of parameters is set by the max_run_fail() control variable (discussed below). If max_run_fail() is set to 1, concurrent model run scheduling is not undertaken. The value X supplied for overdue_resched_fac() must be 1.0 or greater.
4. **++overdue_giveup_fac(2.0)**: If an overdue simulation has already taken X times the average model run time but is still not complete, then the run manager declares the run to have failed and frees the computing agent for another model run based on another set of parameters. The value X supplied for overdue_giveup_fac() must be 1.0 or greater. The default factor is 2.0 for PESTPP-IES, and 100 for PESTPP-GLM.
5. **++overdue_giveup_minutes(1.0E+30)**: This control variable can be used to place a “hard” upper bound on the acceptable model run time. If the duration of a given model run exceeds overdue_giveup_minutes() or overdue_giveup_fac() times the average model run time, the model run stops and the model is marked as a failure. The default value for overdue_giveup_minutes() is 1.0E+30, this effectively disabling it.
6. **++max_run_fail(1)**: Excessive execution time can be deemed as model run failure. Run failure is normally an outcome of model hostility to the set of parameters with which it was provided. However, sometimes it can be caused by external problems that are not the model’s fault, for example operating system quirks or network failure. Hence, the parallel run manager can be instructed to attempt a failed model run again using another agent. The number of attempts is supplied as the value of the max_run_fail() control variable. A value of 1 is used to request no repetition of model runs.

If the *Operation Mode* is “Stochastic (PESTPP-IES)”, the following lines will be appended to the end of the PEST control file in addition to the contents of the text editor.

1. **++ies_num_reals(n)**: This line set the desired number of calibrated realizations to the integer value of n, which is defined in *IES_NUM_REALS* of the Control Data tab.

2. **++ies_subset_size(m)**: This line instructs PESTPP-IES to devote **m** realizations to determining the best Marquardt lambda and line search factor to use during each iteration. The value of **m** is defined in *IES_SUBSET_SIZE* of the Control Data tab.

In addition to the above control variables, you should consider adding the following optional variables to your models. Please consult the PESTPP Users Manual⁷³ and PEST (Doherty, 2016) for details.

1. **++ies_parameter_ensemble(parameter_ensemble_file)**: This instructs PESTPP-IES to load a parameter ensemble file that contains a suite of initial parameter realizations. The RANDPAR3 Utility (that is a part of the PEST Suite) or pyEMU⁷⁵ can be used to generate parameter ensemble files. For models with a great number of parameters, the 64-bit version of the RANDPAR3 Utility (i64RANDPAR3.EXE) should be used. If an initial parameter ensemble is not provided, PESTPP-IES will generate an ensemble based on several assumptions given in the PESTPP Users Manual. Note that this control variable is the same as the “ies_par_en” control variable.
2. **++ies_observation_ensemble(observation_ensemble_file.jcb)**: This control variable instructs PESTPP-IES to load an observation ensemble file that contains random realizations of observations with incorporated measurement noise. Because the ensemble smoother must employ realizations of both random parameter fields and of measurement noise to explore posterior parameter (and hence predictive) uncertainty, a slightly different “calibration dataset” is used for adjustment of each random parameter field. The RANDOBS Utility (that is a part of the PEST Suite) or pyEMU⁷⁵ can be used to generate observation ensemble files. If an observation ensemble file is not provided, PESTPP-IES generates these realizations by using the weights associated with measurements listed in the “observation data” section of the PEST control file as a statistical descriptor of noise stochasticity. It assumes that the weight assigned to each observation is the inverse of the standard deviation of measurement noise associated with that observation⁷⁶. Note that this control variable is the same as the “ies_obs_en” control variable.
3. **++ies_upgrades_in_memory(Boolean)**: This variable sets the flag to hold parameter upgrade ensembles in memory during lambda testing. The default value of Boolean is true. PESTPP-IES will set this flag to False to conserve memory usage if the number of parameters is greater than 100,000. You may consider setting this flag to False, if PESTPP-IES runs out of memory with your model. If the value is set to False, PESTPP-IES essentially stores the candidate parameter ensembles to disk during lambda testing rather than holding them in memory.
4. **++ies_restart_parameter_ensemble(parameter_ensemble_file)** and **++ies_restart_observation_ensemble(observation_ensemble_file)**:

⁷⁵ <https://pyemu.readthedocs.io/en/develop/autoapi/pyemu/index.html>

⁷⁶ https://github.com/usgs/pestpp/blob/master/documentation/pestpp_users_manual.md#916-measurement-noise

During a PESTPP-IES run, a pair of parameter and observation ensemble files after completing an iteration. These control variables can be used to restart/continue a PESTPP-IES run from the end of an iteration using the corresponding pair of ensemble files. These control variables must be used together with the `ies_parameter_ensemble` control variable.

3.9.5.3. Inspect Model Data

Select this to check the geometry of the model and the consistency of the model data. Table 2 shows a list of items that are inspected by PM. The inspection report will be saved in the `modflow_inspection_log.txt` file.


3.9.5.4. Run

This section describes the interface of PM and the major steps that are conducted by PM to build required input files for PEST and the associated flow model.

3.9.5.4.1. Start a Parameter Estimation Process

Select the *Models > PEST > Run with MODFLOW* or *Models > PEST > Run with GSFLOW* menu item to open the Run PEST dialog box (Figure 154), that can be used to build required input files and then start the parameter estimation process. The settings of this dialog box are described below.

1. Table

- The build box** . Prior to running PEST, PM uses the user-specified data to build input files for MODFLOW (or GSFLOW), the PEST control file (`pest.pst`), and other required files for pre- and postprocessors. An input file (and its template file, if applicable) for a package will be built if its build box is checked. If an input file does not exist or the input data has been modified since the last build, its build box will be checked automatically. A template file of a package is needed for PEST, if the package contains adjustable parameters.

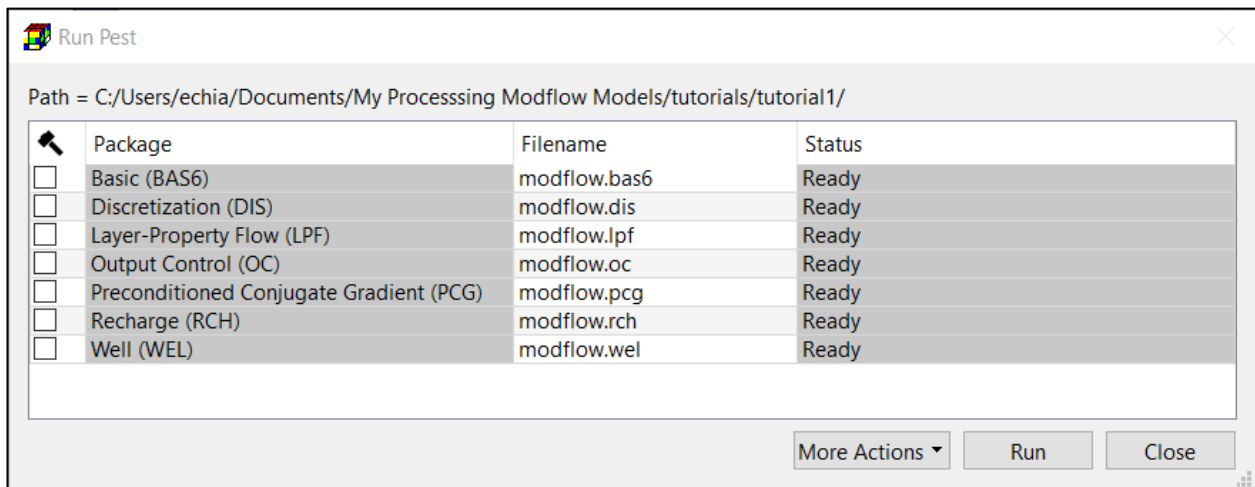


Figure 154. The Run Pest dialog box

- **Package** is the name of the package of the flow model.
- **Filename** is the name of the input file to MODFLOW. The base filename of the corresponding template file to PEST is “pest”, instead of “modflow”.
- **Status** shows the status (such as Ready, Not Exist, etc.) of the input file.

2. More Actions

- **Inspect Model Data:** Select this action to check the geometry of the model and the consistency of the model data. Table 2 shows a list of items that are inspected by PM. The inspection report will be saved in the *modflow_inspection_log.txt* file.
- **Build Input Files:** Select this action to build the input files with the checked build boxes. The name file *modflow.nam* and input file *modflow.lmt* for LMT8 package will also be built. In addition, the PEST control file, all applicable template files, and several batch files (*.bat) are also created for PEST, BeoPEST and/or PESPP-IES. Detailed information is stored in the log files, *modflow_build_log.txt*, *gsflow_build_log.txt*, and *pest_build_log.txt* in the model data folder.
- **Rebuild All Input Files:** Select this action to rebuild all input files, except for existing input files that are marked as read-only.

3. Run:

Click this button to build input files (and their template files) with the checked build boxes and the start the parameter estimation process. Once the required files are built, PM will start the calibration process as described below.

- If the *Operation Mode* of the PEST Settings dialog box (Figure 146) is set to Parameter Estimation, Sensitivity Analysis, Run Model with the Specified PARVAL1 Values, or Regularization, the executable for the PEST run will be selected based on the following rules:
 - If *PEST Version* (in the Preferences dialog box) is “Standard PEST”, the estimation process will be carried out by the executable file of PEST defined in the Preferences dialog box (Figure 37).
 - If *PEST Version* is “BeoPEST”, the estimation process will be carried out by the executable file of BeoPEST defined in the Preferences dialog box (Figure 37).
 - If *PEST Version* is “PESTPP-GLM”, the estimate process will be carried out by the executable file of PESTPP-GLM.

Sections 3.9.5.4.2 and 3.9.5.4.3 describe the major steps to calibrate a deterministic model with PEST, BeoPEST, and PESTPP-GLM.

- If the *Operation Mode* of the PEST Settings dialog box (Figure 146) is set to Stochastic (PESTPP-IES), the executable file of PESTPP-IES defined in the Preferences dialog box (Figure 37) will be used to generate calibration realizations.

Section 3.9.5.4.4 describe the major steps to generate calibrated realizations with PESTPP-IES.

- If the Run PEST dialog box was started by selecting *Models > PEST > Run with MODFLOW*, the flow simulation will be carried out by the executable file associated with the selected MODFLOW Version. If the Run PEST dialog box was started by

selecting *Models > PEST > Run with GSFLOW*, the flow simulation will be carried out by the executable file associated with GSFLOW.

The parameter estimation run progress will be displayed in the PEST Output window. You may terminate the process at any time by clicking on the Terminate button.

3.9.5.4.2. Calibration of a Deterministic Model with PEST

PEST (Doherty, 2016) is used for calibration if “Standard PEST” is selected as the PEST Version in the Preferences dialog box (Figure 37). The major steps that are conducted by PM for calibration of a deterministic model with PEST are illustrated in Figure 155 and described below. These steps are replicated in the windows batch file “*pest.bat*” that is created and saved in the model folder when you click Run, Build Input Files, or Build All Input Files in the Run PEST dialog box. If needed, you can run PEST outside of PM by starting that batch file in a Windows Command Prompt.

1. Preprocess files for the entire calibration process if adjustable parameters include pilot points

- Prepare and execute the batch file *RunPPK2FAC1.bat*. This file calls the *PPK2FAC1* utility application to generate so-called Kriging factors based on the Pilot Points associated with aquifer parameters (such as HK or VK) in individual model layers.
- Prepare and execute the batch file *RunPPKREG1.bat*. This file calls the *PPKREG1* utility application to insert applicable regularization information data to the PEST control file prepared by PM.

2. Start PEST

- Start PEST with the command *pest.exe pest.pst*

3. Calibration Iteration

- For each calibration iteration, PEST will sequentially run as many model instances as needed by using the *modelrun.bat* file. Each model instance involves the following steps.
 - Preprocess input files for an iteration.
 - A parameter is perturbed.
 - If adjustable parameters include pilot points, the batch file *RunFAC2REAL.bat* is called to perform Kriging interpolation using the aquifer parameters (such as HK or VK) associated with pilot points.
 - If adjustable parameters include pilot points, the batch file *RunReal2Matrix.bat* is called to update the model input matrix files (e.g., for HK or VK) based on the Kriging results.
 - If applicable, *FHBAdjust.exe* is called to update the input file to the FHB package with the updated parameter values, including multipliers for specified-flow and addends for specified-head boundaries.
 - Conduct a flow simulation with MODFLOW or GSFLOW.
 - Postprocess output files of MODFLOW or GSFLOW.

- If the *LAYERWEIGHTOPT* box in the Control Data tab is checked, updateweights.exe is called to update the layer weights based on updated HK or T values.
 - mod2obs.exe and layerweight.exe are called to write the simulated head values at the observation wells to an output file.
 - If applicable, streamgage.exe is called to write the simulated streamflow values to an output file.
 - If applicable, lakegage.exe is called to write the simulated lake stage values to an output file.
 - PEST reads the output files based on the instruction files listed in the “* model input/output” section of the PEST control file (pest.pst).
- An updated parameter set (which potentially lowers the objective function) is determined by PEST. The model parameters are then applied to the model input files based on the template files listed in the “* model input/output” section of the PEST control file (pest.pst).
 - Repeat the iteration until the calibration criteria or stop criteria are met.
- 4. Final Run with the estimated parameters.** If LASTRUN (in the Control Data tab of the PEST Settings dialog box) is checked, PEST will conduct the final model run with the set of estimated parameters that resulted in the lowest objective function.

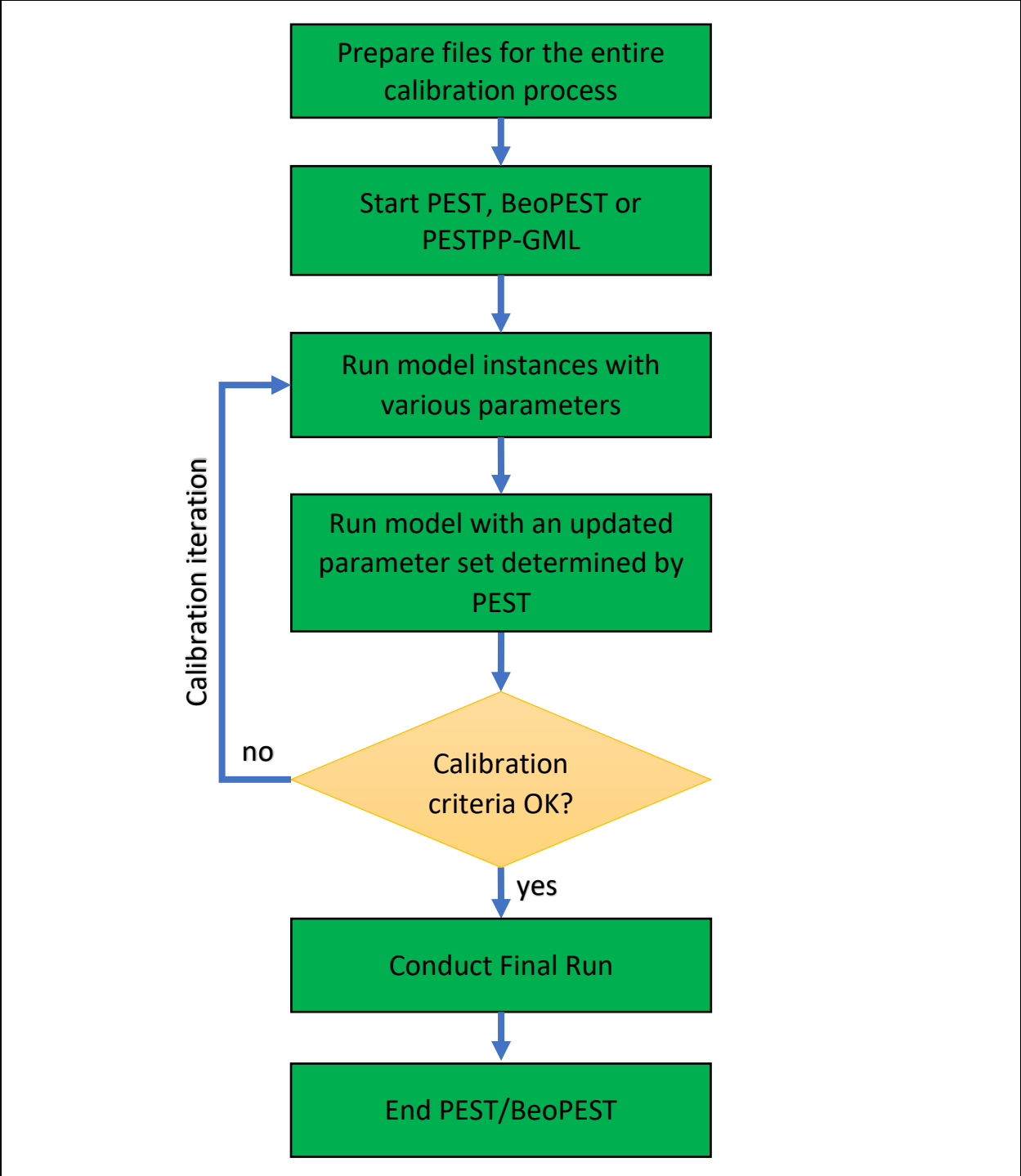


Figure 155. Flow Chart of model calibration with PEST, BeoPEST or PESTPP-GLM

3.9.5.4.3. Calibration of a Deterministic Model with BeoPEST or PESTPP-GLM

BeoPEST or PESTPP-GLM is used for calibration if the corresponding code is selected as the PEST Version in the Preferences dialog box (Figure 37). The major steps that are conducted by PM for calibration of a deterministic model with these codes are similar to those for Standard PEST. However, several additional steps are required to enable available computing agents of BeoPEST or PESTPP-GLM to run parallelly.

The detailed steps for running BeoPEST are given below. These steps are replicated in the windows batch file “BeoPEST.bat” that is created and saved in the model folder when you click Run, Build Input Files, or Build All Input Files in the Run PEST dialog box. If needed, you can run BeoPEST outside of PM by opening a Windows Command Prompt, making the model folder as the current folder with the CD command, typing BeoPEST.bat and then pressing enter. You can terminate the BeoPEST run by pressing Ctrl+C, and then closing the Windows Command Prompt.

The steps for running PESTPP-GLM are similar to those for BeoPEST. The batch file for running PESTPP-GLM in a Windows Command Prompt is “PestppGLM.bat”.

1. Preprocess files for the entire calibration process if adjustable parameters include pilot points

- Prepare and execute the batch file RunPPK2FAC1.bat. This file calls the PPK2FAC1 utility application to generate so-called Kriging factors based on the Pilot Points associated with aquifer parameters (such as HK or VK) in individual model layers.
- Prepare and execute the batch file RunPPKREG1.bat. This file calls the PPKREG1 utility application to insert applicable regularization information data to the PEST control file prepared by PM.

2. Replicate model files to folders of computing agents

- Prepare the batch file beopest_agent.bat that contains the following command. This batch file is used to start a BeoPEST computing agent.

```
beopest64.exe pest.pst /h 127.0.0.1:4004 >pest.log
```
- Create separate folders for computing agents and replicate model input files and the beopest_agent.bat file to those folders.
- For advanced users: The IP address (127.0.0.1) in the above command works only if the computing agents and the Run Manager run on the same computer, and that is the default behavior when you start BeoPEST from PM. If you start BeoPEST with BeoPEST.BAT, you can run additional computing agents on other computers on the same network by (1) replicating the folder and files of a computing agent (e.g., PestAgent1), (2) updating its beopest_agent.bat file by replacing 127.0.0.1 with the IP address of the computer where the Run Manager is located, and then (3) starting that updated pestppies_agent.bat file in a Windows command prompt. To start many computing agents, you might want to create a batch file by using the sections

“Replicate files of PestAgent1 to other computing agents” and “Start all computing agents” of the BeoPEST.BAT file.

3. Start BeoPEST Computing Agents and Run Manager

- Start all computing agents by executing the following command in the folder of each computing agent:

Start /B /MIN beopest_agent.bat

- Start the BeoPEST Run Manager by executing the following command in the model folder:

beopest64.exe pest.pst /L /H :4004 /p1

4. Calibration Iteration

- For each calibration iteration, BeoPEST will run model instances on available computing agents by using the modelrun.bat file. The number of available computing agents is defined in the Preferences dialog box (Section 3.5.12). Each model instance involves the following steps.
 - Preprocess input files.
 - A parameter is perturbed.
 - If adjustable parameters include pilot points, the batch file RunFAC2REAL.bat is called to perform Kriging interpolation using the aquifer parameters (such as HK or VK) associated with pilot points.
 - If adjustable parameters include pilot points, the batch file RunReal2Matrix.bat is called to update the model input matrix files (e.g., for HK or VK) based on the Kriging results.
 - If applicable, FHBAdjust.exe is called to update the input file to the FHB package with the updated parameter values, including multipliers for specified-flow and addends for specified-head boundaries.
 - Conduct a flow simulation with MODFLOW or GSFLOW.
 - Postprocess output files of MODFLOW or GSFLOW.
 - If the *LAYERWEIGHTOPT* box in the Control Data tab is checked, updateweights.exe is called to update the layer weights based on updated HK or T values.
 - mod2obs.exe and layerweight.exe are called to write the simulated head values at the observation wells to an output file.
 - If applicable, streamgage.exe is called to write the simulated streamflow values to an output file.
 - If applicable, lakegage.exe is called to write the simulated lake stage values to an output file.
 - PEST reads the output files based on the instruction files listed in the “* model input/output” section of the PEST control file (pest.pst).

- An updated parameter set (which potentially lowers the objective function) is determined by PEST. The model parameters are then applied to the model input files based on the template files listed in the “* model input/output” section of the PEST control file (pest.pst).
 - Repeat the iteration until the calibration criteria or stop criteria are met.
- 5. Final Run with the estimated parameters.** The run manager of BeoPEST will conduct the final model run with the set of estimated parameters that resulted in the lowest objective function.

3.9.5.4.4. Generate Calibrated Realizations with PESTPP-IES

The calibration process with PESTPP-IES results in several calibrated realizations – each with a set of parameters that aims to calibrate the model. The major steps that are conducted by PM to generate calibrated model realizations with PESTPP-IES are illustrated in Figure 156 and described below. These steps are replicated in the windows batch file *PestppIES.bat* that is created and saved in the model folder when you click Run, Build Input Files, or Build All Input Files in the Run PEST dialog box. If needed, you can run PESTPP-IES outside of PM by opening a Windows Command Prompt, making the model folder as the current folder with the “CD” command, then starting the *PestppIES.bat* batch file. You can terminate the *PestppIES.bat* file and thus the PESTPP-IES run by pressing Ctrl+C, and then closing the Windows Command Prompt. If you need to add additional packages that are not supported by PM, you can add them to the model by editing the modflow.nam file and making sure that those packages refer to consistent unit numbers for input and output files listed in the modflow.nam file.

1. Preprocess files for the entire calibration process if adjustable parameters include pilot points

- If adjustable parameters include pilot points, execute the batch file RunPPK2FAC1.bat. This file calls the PPK2FAC1 utility application to generate so-called Kriging factors based on the Pilot Points associated with aquifer parameters (such as HK or VK) in individual model layers.
- Execute the batch file RunPPCOV.bat. This file calls the PPCOV utility (Doherty, 2018) to generate covariance matrix files based on the Pilot Points associated with aquifer parameters (such as HK or VK) in individual model layers. A required parameter uncertainty file is comprised of the generated covariance matrix files.

2. Replicate model files to folders of computing agents

- Prepare the batch file pestppies_agent.bat that contains the following command. This batch file is used to start a computing agent that will work together with the PESTPP-IES Run Manager (see below).

```
pestpp-ies.exe pest.pst /h 127.0.0.1:4004 >pestppies.log
```

- Create separate folders for computing agents and replicate model input files and the pestppies_agent.bat file to those folders.
- For advanced users: The IP address (127.0.0.1) in the above command works only if the computing agents and the Run Manager run on the same computer, and that is the default behavior when you start PESTPP-IES from PM. If you start PESTPP-IES with PestppIES.BAT, you can run additional computing agents on other computers on the same network by (1) replicating the folder and files of a computing agent (e.g., PestAgent1), (2) updating its pestppies_agent.bat file by replacing 127.0.0.1 with the IP address of the computer where the Run Manager is located, and then (3) starting that updated pestppies_agent.bat file in a Windows command prompt. To start many computing agents, you might want to create a batch file by using the sections “Replicate files of PestAgent1 to other computing agents” and “Start all computing agents” of the PestppIES.BAT file.

3. Start PESTPP-IES Computing Agents and Run Manager

- Start all computing agents by executing the following command in the folder of each computing agent:

```
Start /B /MIN pestppies_agent.bat
```

- Start the PESTPP-IES Run Manager by executing the following command in the model folder:

```
pestpp-ies.exe pest.pst /h :4004
```

4. Calibration Iteration

- For each calibration iteration, PESTPP-IES will run model instances on available computing agents by using the modelrun.bat file. The number of available computing agents is defined in in the Preferences dialog box (Section 3.5.12). Each model instance involves the following steps.
 - Preprocess input files.
 - Parameters are updated by PESTPP-IES.
 - If adjustable parameters include pilot points, the batch file RunFAC2REAL.bat is called to perform Kriging interpolation using the aquifer parameters (such as HK or VK) associated with pilot points.
 - If adjustable parameters include pilot points, the batch file RunReal2Matrix.bat is called to update the model input matrix files (e.g., for HK or VK) based on the Kriging results.
 - If applicable, FHBAAdjust.exe is called to update the input file to the FHB package with the updated parameter values, including multipliers for specified-flow and addends for specified-head boundaries.
 - Conduct a flow simulation with MODFLOW or GSFLOW.
 - Postprocess output files of MODFLOW or GSFLOW.

- If the *LAYERWEIGHTOPT* box in the Control Data tab is checked, updateweights.exe is called to update the layer weights based on updated HK or T values.
- mod2obs.exe and layerweight.exe are called to write the simulated head values at the observation wells to an output file.
- If applicable, streamgage.exe is called to write the simulated streamflow values to an output file.
- If applicable, lakegage.exe is called to write the simulated lake stage values to an output file.
- PESTPP-IES reads the output files based on the instruction files listed in the “* model input/output” section of the PEST control file (pest.pst).
- An updated parameter set is determined. The model parameters are then applied to the model input files based on the template files listed in the “* model input/output” section of the PEST control file (pest.pst).
- An updated ensemble of parameter sets is determined. Repeat the iteration until the calibration criteria or stop criteria are met. See next Section for the steps to run a model using estimated parameter values.

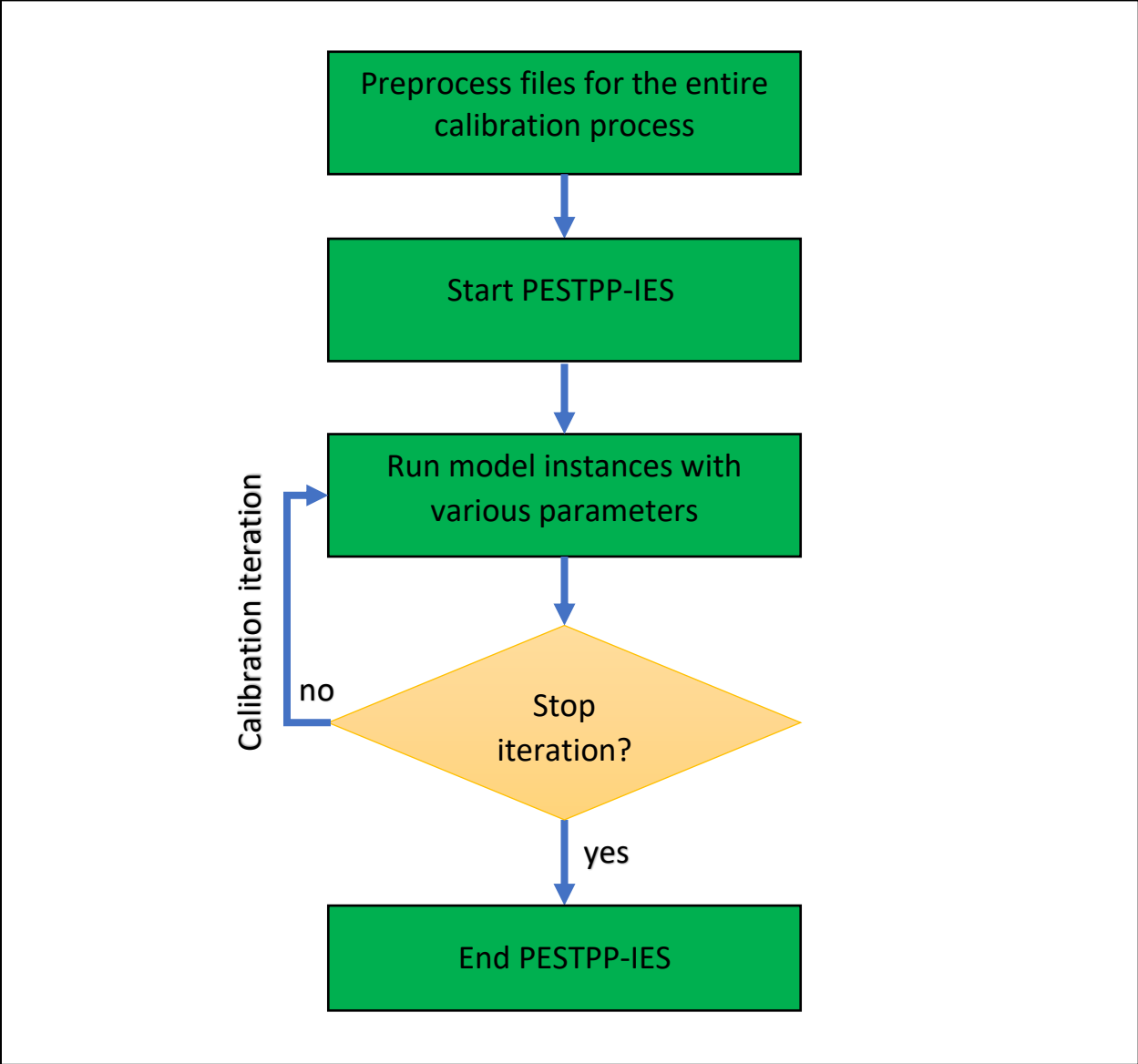


Figure 156. Flow Chart of model calibration with PESTPP-IES

3.9.5.4.5. Running a Model with Estimated Parameter Values

You can run a model with a set of estimated parameter values⁷⁷ by (1) importing them to the PEST Settings dialog box (Figure 146) with its *More Actions > Update PARVAL1*, (2) applying parameter values to model cells with *More Actions > Apply PARVAL1*, and then (3) starting the flow simulation with *Models > MODFLOW > Run* or *Models > GSFLOW > Run*, whichever is applicable to your model. During the execution of the *Apply PARVAL1* action, the estimated parameter values are applied to model cells and if the *LAYERWEIGHTOPT* box in the Control Data tab is checked, the layer-weight values of multi-layer observations in the Observation dialog box (Section 3.6.3) are updated based on estimated HK or T parameter values.

Alternatively, you can run a model with a set of estimated parameter values⁷⁸ by (1) importing them to the PEST Settings dialog box (Figure 146) with its *More Actions > Update PARVAL1*, (2) changing the operation mode of PEST to “Run Model with the Specified PARVAL1 Values,” and then (3) starting PEST with *Models > PEST > Run with MODFLOW* or *Models > PEST > Run with GSFLOW*, whichever is applicable to your model. Prior to executing MODFLOW or GSFLOW, the estimated parameter values are applied to model cells in the input files of various packages by using the information contained in the batch file *modelrun.bat* and the *model input/output* section of the *pest.pst* file. If the *LAYERWEIGHTOPT* box in the Control Data tab is checked, the *modelrun.bat* will also contain a command line to run *updateweights.exe*, which will update the layer weight values based on HK values and save the updated weight values in the *pest.headboreinfo* file. That file is then used by *layerweight.exe* to calculate layer-weighted head values and save them in the *pest.headoutput* file. And, finally, PEST reports statistics based on the *pest.headoutput* file. During the entire process, the layer weight values of multi-layer observations in the Observation dialog box (Section 3.6.3) are not updated. Nor are the cell values in the model under the PM interface updated. Thus, if you intend to display head or drawdown values in the Scatter Chart (Section 3.9.1.6.3) or Line Chart (Section 3.9.1.6.4), it is strongly recommended to use *More Actions > Apply PARVAL1* of the PEST Settings dialog box (Figure 146) so that the updated layer weights are used to generate the layer-weighted head or drawdown time series.

In addition, several utility programs of the PEST suite (Doherty, 2018) can be used to run models with the estimated parameters in the Windows Command Prompt⁷⁹ (i.e., the DOS box). The

⁷⁷ PEST usually stores a set of estimated parameter values in a PEST parameter value file that usually has the file extension “.par”. A PEST parameter value file can be imported to the PEST Settings dialog box by clicking its *More Actions > Update* button.

⁷⁸ PEST usually stores a set of estimated parameter values in a PEST parameter value file that usually has the file extension “.par”. A PEST parameter value file can be imported to the PEST Settings dialog box by clicking its *More Actions > Update* button.

⁷⁹ You should change the current directory of the Windows Command Prompt to the model data folder, and then run PEST and its utility programs from there. PEST and its utility programs are installed in the \engine\pest subfolder under the PM application folder. You need to make sure that the path of the \engine\pest subfolder is included in

following paragraphs outline the process to run models with the results from PEST and PESTPP-IES.

PEST

The PARREP utility can be used together with PEST to run a model with estimated parameter values. The PARREP utility replaces initial parameter values in an existing PEST control file by a set of values stored in a PEST parameter value file, and then stores the results in a new PEST control file. PEST can use that file to run the model.

The following command, for example, replaces the initial parameter values of the existing PEST control file *pest.pst* with the contents of the PEST parameter value file *pest.par*, sets the value of NOPTMAX to 0, and then save the results in the new PEST control file *OneRun.pst*. Setting NOPTMAX to 0 instructs PEST to conduct only one model run with the parameter values given in the control file, and then terminate execution after that model run.

```
parrep pest.par pest.pst OneRun.pst 0
```

Once the new PEST control file is created. You can start PEST by entering the command ***pest OneRun.pst*** in the Windows Command Prompt.

PESTPP-IES

All output files written by PESTPP-IES use the same filename base as the PEST control file (by default, PM uses the filename *pest.pst*). The *pest.phi.actual.csv* file contains the objective function values based on the observed values and simulated values of each parameter realization after completion of every iteration. Some of the output files are JCB files when the line “++ies_save_binary(true)” is included in the PEST control file. For example, the *pest.N.par.jcb* files contain the parameter realizations after the completion of the Nth iteration and the *pest.N.obs.jcb* files contain the observation realizations used in the Nth iteration. You can use the JCB2CSV utility to convert a JCB file to a CSV file, if you want to inspect its contents.

The JCB2PAR utility can be used to extract a parameter realization from a JCB file. The following command, for example, extracts the 3rd parameter realization from the *pest.10.par.jcb* file and saves the extracted data to the PEST parameter value file *realization10_3.par*.

```
jcb2par pest.10.par.jcb 3 realization10_3.par
```

The PARPEP utility can be used to prepare for a single model run with the extracted PEST parameter value file. The following command, for example, creates the new PEST control file *realization10_3.pst* based on the original PEST control file *pest.pst* and the PEST parameter value file *realization10_3.par*.

the PATH variable of Windows. Alternatively, you could copy the executables of PEST and its utility programs to the model data folder.

```
parrep realization10_3.par pest.pst realization10_3.pst 0
```

Once the new PEST control file is created and saved in the model folder where other model files are stored. You can start PEST by entering the command ***pest realization10_3.pst*** in the Windows Command Prompt.

3.9.5.5. Run PESTCHEK

Once the model input files are generated (through the Run PEST dialog box), you can use PESTCHEK to check for the correctness and consistency of the PEST control file built by PM. Running PESTCHEK is normally not necessary. However, PESTCHEK may provide additional advice beyond data checking.

3.9.5.6. View

The View menu allows you to display several output files of PEST or BeoPEST. These files are described below. For detailed discussions, refer to Part I of the PEST Manual (Doherty, 2016). Note that this menu does not apply to PESTPP-IES as it generates result files of multiple realizations, see Section 3.9.5.4 for details.

3.9.5.6.1. Run Record File

As PEST executes, it writes a detailed record of the parameter estimation process to the run record file *pest.rec*. If the parameters and/or observations are large in number this file can get very long. In that case, you should uncheck VERBOSEREC box in the Control Data Tab of the PEST Settings dialog box.

3.9.5.6.2. Estimated Parameter Values

At the end optimization-iteration, PEST writes the best parameter set achieved so far (i.e., the set for which the objective function is lowest) to the parameter file *pest.par*. The first line of the file contains the values for the character variables PRECIS and DPOINT, which were used in the PEST control file. Then follows a line for each parameter, each line containing a parameter name, its current value and the values of the SCALE and OFFSET variables for that parameter. Using values from intermediate parameter-estimation iterations, that are likely to be closer to the optimal parameter values, often reduces execution time.

3.9.5.6.3. Residuals

On completion of parameter estimation process, PEST writes a “residuals file” tabulating observation names, the groups to which observations belong, measured and modelled observation values, differences between these two (i.e., residuals), measured and modelled observation values multiplied by respective weights, weighted residuals, measurement standard deviations and “natural weights”. This file can be readily imported into a spreadsheet for various types of analysis and plotting. The name of the residuals file is *pest.res*.

3.9.5.6.4. Interim Residuals

At the beginning of each iteration PEST writes the interim residuals file `pest.rei`. It contains six columns of data. The first two columns cite observation names and group names; then follow observation values and their current model-generated counterparts. Current residuals and weights follow that in subsequent columns.

3.9.5.6.5. Composite Parameter Sensitivities

Immediately after it calculates the Jacobian matrix, PEST writes composite parameter sensitivities to the parameter sensitivity file `pest.sen`. The information contained in this file allows you to track variations in the composite sensitivity of each parameter through the parameter estimation process.

3.9.5.6.6. Composite Observation Sensitivities

The Observation Sensitivity file `pest.seo` contains all observation values and corresponding model-calculated values, as well as composite sensitivities for all observations. The composite sensitivity of an observation is a measure of the sensitivity of that observation to all parameters involved in the parameter estimation process. A high value of composite observation sensitivity normally indicates that an observation is particularly crucial to the inversion process.

3.9.5.6.7. SVD Output File

When SVD is activated, PEST writes the SVD output file `pest.svd` in addition to other output files. The SVD output file can become very large and not all the information recorded in this file is always worth reading. However sometimes an inspection of singular values can assist in understanding the degree to which an inverse problem is ill-posed.

3.9.5.6.8. Scatter Chart

To display the scatter chart based on the head data, select *Scatter Chart > Head*. To display the scatter chart based on the drawdown data, select *Scatter Chart > Drawdown*. For details, see Section 3.9.1.6.3.

3.9.5.6.9. Line Chart

To display the line chart based on the head data, select *Line Chart > Head*. To display the line chart based on the drawdown data, select *Line Chart > Drawdown*. For details, see Section 3.9.1.6.4.

Chapter 4. Visualization and Postprocessing

This chapter describes how to visualize and postprocess the input/output data of the active model. A model in the TOC contains two major groups: **Input Data** and **Visualization**. The Input Data group allows you to adjust the visualization settings for the model's input data, including model grid, flow packages, and observation points. The Visualization group includes various types of items, such as input parameter contours, simulation result contours, flow vectors, and pathlines from MODPATH instances. Additionally, scatter charts and line charts can be used to display measured and simulated time series at observation points. For more details, refer to the View menus in Sections 3.9.1.5, 3.9.3.8, and 3.9.5.6.

As mentioned in Chapter 1, each item in the TOC represents a map layer that contains a visual representation of its corresponding data. When you click on an item in the TOC, it becomes the active layer. The available visualization settings for the active layer are displayed in the **Settings window** and can be modified accordingly. These settings allow you to control various aspects of how the data is visualized on the map, including color schemes, transparency, and other visual elements related to the active layer's data.

4.1. Input Data

4.1.1. Model Grid

As shown in Figure 157, the display settings of the model grid are organized in several groups based on the nature of the data, such as no-flow cells, variable-head cells, model grid, etc. These groups are described below. By right-clicking on the *Model Grid* item and selecting *Export to Shapefile*, you can export the shapefiles of the visible groups to a zip file.

1. General

- **Display Layer:** This is the layer number of the model grid. When multiple items are displayed on the Map Viewer, the items with smaller layer numbers are displayed on top of the items with larger layer numbers.

2. Model Bounding Box

- **Visible:** Check this box to turn on the display of the model bounding box encompassing all cells of the model.
- **Opacity:** The model box is if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the model box.
- **Color:** Use this dropdown box to select a color for the model box.

3. Grid Lines

- **Drawing Options**
 - **Do Not Draw:** The grid lines will not be displayed.

- **All Cells:** The grid lines of all cell types will be displayed.
- **Variable-Head Cells Only:** The grid lines of variable-head cells will be displayed.
- **No-Flow Cells Only:** The grid lines of no-flow cells will be displayed.
- **Opacity:** The grid lines are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the grid lines.
- **Color:** Use this dropdown box to select a color for the grid lines.

4. Variable-head Cells, No-flow Cells, Dry Cells⁸⁰, Constant-head Cells, Active Concentration Cells, Inactive Concentration Cells, Fixed Concentration Cells: The available settings of these groups are the same. Use them to adjust the display settings of the respective cell types.

- **Visible:** Check this box to display the cells.
- **Opacity:** The cells or their perimeter is opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Perimeter Line Width** is the line width (in pixels) that is used to draw the perimeter of the cells. This value is used only if the Drawing Option (see below) is *Perimeter*.
- **Color:** Use this dropdown box to select a color for the cells or their perimeter.
- **Drawing Options**

- **Color Filled:** If this option is selected, all cells will be filled with the color selected by the Color dropdown box.
- **Perimeter:** If this option is selected, the perimeter of the cells will be drawn with the line width and color defined in *Perimeter Line Width* and *Color*.

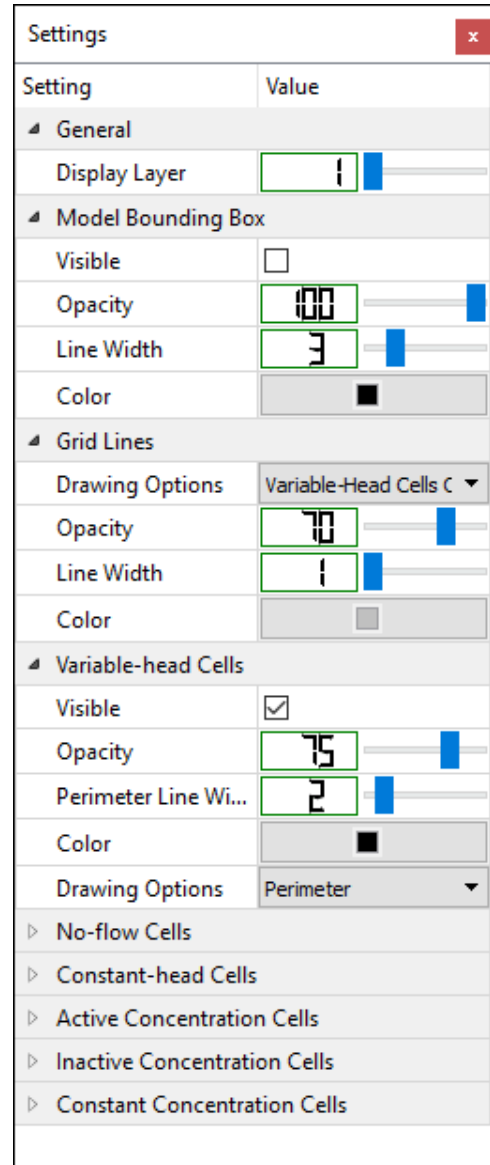


Figure 157. The Settings window for model grid

⁸⁰ Dry Cells are the model cells that went dry during a flow simulation. The hydraulic head result file modflow.head file is required for Dry Cells to work.

4.1.2. Packages

Each item in this group corresponds to a MODFLOW package. By right-clicking on the *Packages* item and selecting *Export to Shapefile*, you can export the shapefiles of the visible packages **Error! Bookmark not defined.** to a zip file. By right-clicking on an item in the Packages group, you have access to the following options:

1. **Export to Shapefile****Error! Bookmark not defined.**: this will export the model cells of the package to a shapefile.
2. **Remove Package**⁸¹: this option removes the package and erases its data.
3. **Deactivate Package**⁸¹: this option deactivates the package but retains its data. A deactivated package may be activated by selecting the corresponding item in the *Models > Modflow > Packages* menu.

Each package has a Display Setting group in the Settings window. The following display settings are common among packages.

1. **Display Layer**: This is the layer number of the package. When multiple map layers are displayed, the layers with smaller layer numbers are displayed on top of the layers with larger layer numbers.
2. **Opacity**: The cells of the package is opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
3. **Color**: Use this dropdown box to select a color for the cells of the package. Note that some packages have several color boxes. For example, the Well Package has separate color boxes for pumping, injection, and inactive wells.
4. **Highlight Parameter**: Within the Data Editor, you can double-click on a cell to highlight all other cells that have the same cell value as the clicked cell. The cell values of the selected Highlight Parameter will be used for this purpose.

4.1.3. Observation Points

By right-clicking on an item in this group, you can display a popup menu that allows you to edit, import, or export observation points and data. For details, see Section 3.6.3. The visualization settings of the observation points (Figure 158) are as follows:

1. General

- **Display Layer**: This is the layer number of the Observation Points. When multiple items are displayed, the items with smaller layer numbers are displayed on top of the items with larger layer numbers.

2. Observation Point

- **Draggable** determines whether the observation points may be dragged by the mouse.

⁸¹ This option is available only when the package is not being edited.

- **Symbol Opacity:** The observation points are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the observation points.
- **Symbol Size** defines the size of the observation point. Click the [...] button to reset to the default size.

3. **Labels** defines the properties of observation point labels.

- **Visible:** Check this box to display the labels of the observation points.
- **Opacity:** The labels are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the labels.
- **Font Size** defines the size of the labels. Click the [...] button to reset to the default size.
- **Color:** Use this dropdown box to select a color for the labels.

4.2. Visualization

4.2.1. Input Parameter Contours

4.2.1.1. Create Contour Maps Based on Input Parameters

You can create contour maps based on the cell-by-cell values of model input parameters by right-clicking on Input Parameter Contours on the TOC, select *Add* from a popup menu, and then select an input parameter from the Select Data dialog box. The following input parameters are supported:

1. Data entered through the Discretization and Parameters menus.
2. MODFLOW: Evapotranspiration (EVT), Evapotranspiration Segments (ETS), Recharge (RCH), as well as Subsidence and Aquifer-Compaction Packages.
3. MT3D: Dispersion (DSP), Reaction (RCT), and Sink & Source Mixing (SSM) Packages.
4. SEAWAT: Prescribed fluid density and prescribed dynamic viscosity.

After adding a Contour Map item to the TOC, you can right-click on the item to access the following functions:

1. Show All Items: the contour map contains several items, one for each contour level. Select this to display all contour levels.
2. Hide All Items: Select this to hide all contour levels.

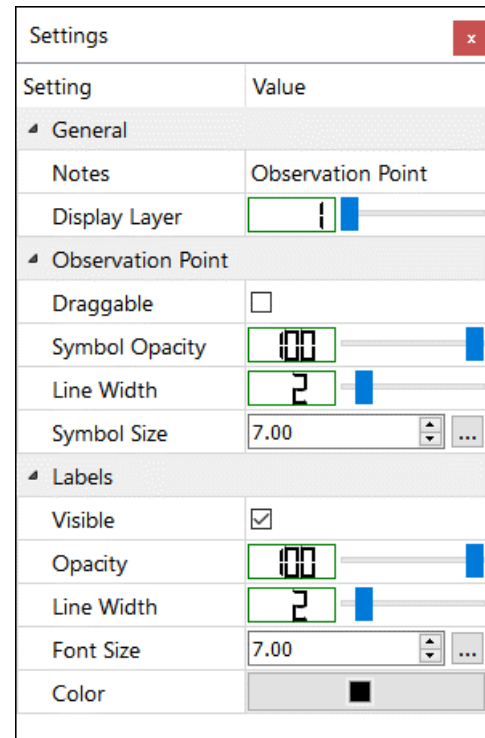


Figure 158. The Settings window for observation points

3. Refresh: Select this to update the contour map. This item is displayed only if the contour map is based on the data that is being edited.
4. Export to Shapefile: Select this to display the Save As dialog box where you can select a shapefile (with the extension .shp) for storing the contour map.
5. Export Matrix Data: Select this to open the Export Data dialog box (see Figure 48 on page 85 for details).
6. Copy Current Layer: Select this to open the Destination dialog box that allows you to copy the matrix data of the current layer to the corresponding layer of the selected destination. If the selected destination is a Transient Sandbox, the matrix data will be saved to the current stress period of the selected Transient Sandbox.
7. Copy All Layers: Select this to open the Destination dialog box that allows you to copy the matrix data of all layers to the selected destination. If the selected destination is a Transient Sandbox, the matrix data will be saved to the current stress period of the selected Transient Sandbox.
8. Remove Item: Select this to remove the Contour Map item.

4.2.1.2. Customize Contour Maps

The visualization settings of the input parameter contour map are as follows:

1. General (Figure 159)

- **Name:** The text string is the name of the Contour Map displayed on the TOC.
- **Notes:** The text string can be used for notes.
- **Reference** may be either None or one of the following parameters: Sand Box 1, Sand Box 2, Sand Box 3, Starting Head, Top of Layer, and Bottom of Layer.
 - If None is selected, the cell values are used for constructing the contour map,
 - Contours will not be created for the areas with the reference value equals to -1E20. You can crop out an area by assigning -1E20 to the cells within that area,
 - If the *Invert Reference* box is checked, the results of “reference - cell values” are used for constructing the contour map. If the *Invert Reference* box is cleared, the results of “cell values – reference” are used for constructing the contour map. You can, for example, create a contour map of layer thickness by selecting Top of Layer as the input parameter and using Bottom of Layer as the Reference.

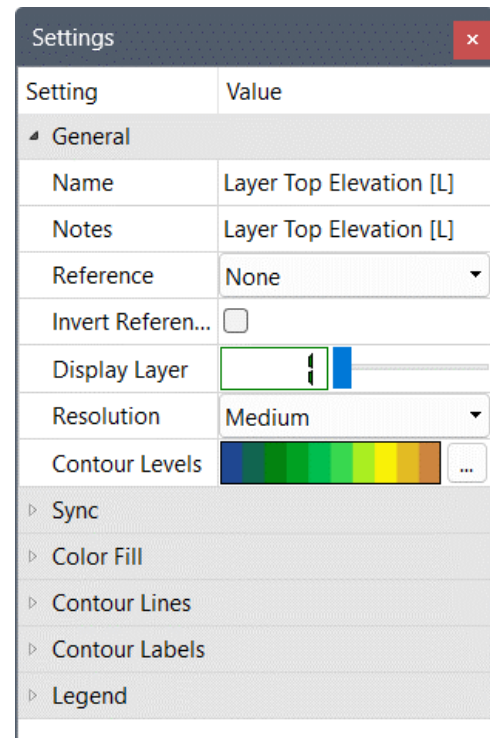



Figure 159. The General group of the Settings window for Contour Maps

- **Invert Reference:** Check the box to invert the reference (see *Reference* above).
- **Display Layer:** This is the layer number of the contour map. When multiple items are displayed, the items with smaller layer numbers are displayed on top of the items with larger layer numbers.
- **Resolution** determines the maximum number of values that are used for contouring. The available options with the corresponding maximum number of values are listed below. It is recommended to use a lower resolution when trying out various settings.
 - **Native:** Use the native model resolution.
 - **Highest:** 150,000.
 - **High:** 112,500
 - **Medium:** 75,000
 - **Low:** 37,500.
 - **Lowest:** 5,000.

Before a matrix of the input parameter values (the matrix) is used for contouring, the matrix may be refined or down-sampled to match the selected resolution according to the following rules.

- For contours on the Map Viewer, the following rules apply to the matrix:
 - If a model has only one row or one column, the matrix at its native resolution is used.
 - If a model has more than one row and one column, the matrix is either refined (by inserting values between cells and estimating those values by linear interpolation based on nearby cell values) or down-sampled to match the selected resolution.
- For contours on the Cross-section window:
 - If a model has only one layer, the matrix at its native resolution is used.
 - If a model has more than one layer,
 - if the native resolution is selected or if the selected resolution is lower than the native resolution, the matrix at its native resolution is used. Contours will not be displayed in the parts of the model where only one layer is active.
 - if the selected resolution is higher than the native resolution, the matrix is refined by inserting values on the top/bottom of model and between cell centers. Those values are estimated by linear interpolation using up to four values of surrounding cells. The inserted values on the top or bottom of the model are estimated by linear interpolation using up to two values of surrounding cells in the top or bottom layers, respectively.

- **Contour Levels:** The contour levels can be defined by clicking on the  button to open the Classes dialog box (Figure 160).

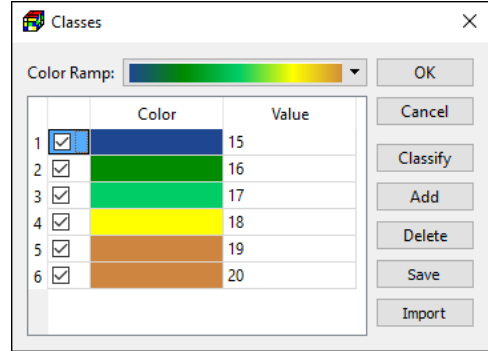


Figure 160. The Classes dialog box

- **Color-Mapping Table:** The table displays the colors and corresponding parameter values. The default values of the table are based on the minimum and maximum values of all cells. Cell values, that are equal to -1E20, less than the first row or greater than the last row, are not used for contouring. When displaying color-filled contours, the color of a row is used to fill the area between the contour levels of the given row and the next row.
- **Color Ramp:** Select a predefined color ramp from this dropdown box to assign graduated colors to the color-mapping table.
- **Classify:** Click this button open the Classify dialog box (Figure 161) that is used to define the range of values and number of classes. If *Min. Value* and *Max. Value* are both positive, the option of *Use logarithmic scale between Min and Max Values* is available. Click the Default button to reset *Min. Value* and *Max. Value* to the values found in the data of the current layer.
- **Add** is used to add a contour level to the table. Click this button to open the Add Value dialog box, where a new contour can be entered.
- **Delete:** To delete a contour level, first click on the desired row on the table, then click this button.
- **Save:** Click this button to save the color-mapping table in an XML file.
- **Import:** Click this button to import a saved XML file to the color-mapping table.

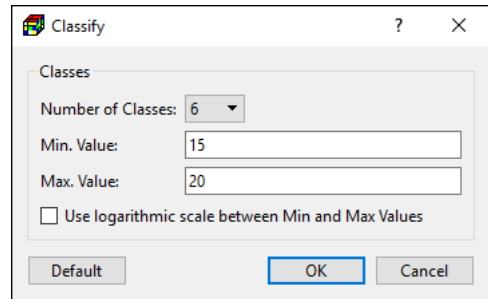


Figure 161. The Classify dialog box

2. **Sync** (Figure 162): This group controls what data is used to generate contours.

- **Mode**

- **Auto:** The stress period, layer, species controls are deactivated and synced with the corresponding controls on the toolbar. The contour map will be updated accordingly, whenever any of the controls on the toolbar is changed or if the values of the input parameter are modified by using *Tools > Data > Modify* or *Tools > Data > Import*.

- **Lock:** The stress period, layer, species controls are deactivated and locked at their current settings. The contour map will never change, even if the values of the input parameter are modified.

- **Manual:** The stress period, layer, species controls are activated when applicable. The contour map will be updated accordingly, whenever any of these controls is changed or if the values of the input parameter are modified by using *Tools > Data > Modify* or *Tools > Data > Import*.

- **Stress Period:** For time-independent model data, such as HK or T, this control is deactivated and is not used. For time-dependent model data, such as Recharge Flux, the data of the selected stress period is displayed. For transient sand boxes the following rules apply:

- If the Mode is Auto, contour maps are created based on the matrix values that are linearly interpolated from the nearby stress periods to the time point at the end of the selected stress period and time step (on the toolbar). When running an animation process, the contour map is created for individual time frames.

- If the Mode is Manual, contour maps are created based on the matrix values that are linearly interpolated from the nearby stress periods to the time point at the end of the stress period defined by this control.

- **Layer:** For layer-dependent model data (such as HK), the data of the selected layer is displayed. For layer-independent model data (such as Maximum EVT Flux), this control is deactivated and is not used.

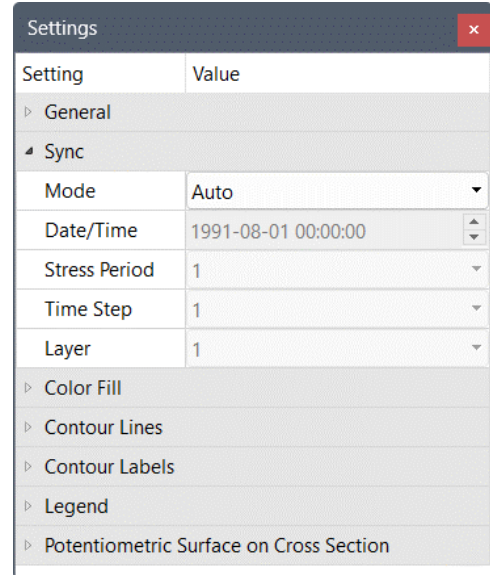


Figure 162. The Sync group of the Settings window for Contour Maps

- **Species:** For species-dependent model data (such as initial concentration), the data of the selected layer is displayed. For species-independent model data (such as HK), this control is deactivated and is not used.

3. Color Fill (Figure 163)

- **Visible:** Check this box to fill the contours with the colors defined in the color-mapping table.
- **Opacity:** The color-fill is opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.

4. Contour Lines (Figure 163)

- **Visible:** Check this box to display the contour lines.
- **Opacity:** The contour lines are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the contour lines.
- **Color:** Use this dropdown box to select a color for the contour lines.

5. Contour Labels (Figure 163)

- **Visible:** Check this box to display the contour labels.
- **Opacity:** The contour labels are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the contour labels.
- **Distance:** This is the distance (in the model length unit) between contour labels. A smaller number yields more labels.
- **Font Size:** This is the font size (in the model length unit) of the contour labels.
- **Color:** Use this dropdown box to select a color for the contour labels.
- **Format**
 - **Decimal Notation** formats values in the form [-]9.9.
 - **Scientific Notation** formats values in the form [-]9.9E[±]999.
 - **Compact Notation** uses the decimal or scientific notation, whichever is the most concise.
- **Precision:**

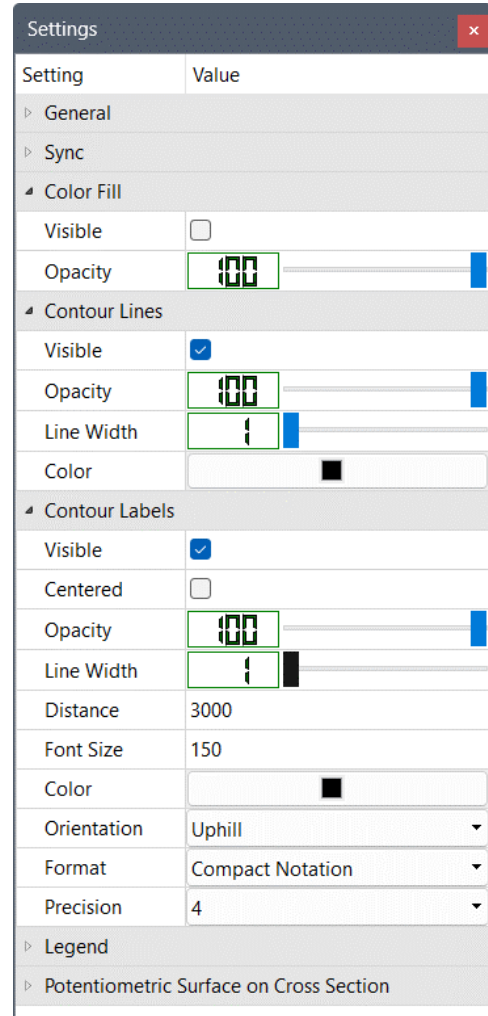


Figure 163. The Color Fill, Contour Lines, and Contour Labels groups of the Settings window for Contour Maps

- For Decimal and Scientific Notations: maximum number of digits after the decimal point
- For Compact Notation: maximum number of significant digits.

6. Legend (Figure 164)

- **Visible:** Check this box to display the on-map legend.
- **Enable Interaction:** Check this box to allow for resizing and moving the legend with the mouse.
- **Opacity:** The legend is opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Width:** This defines the relative width of the legend. The larger the number the wider is the legend.
- **Reset Legend:** Click the Reset button to reset the legend to its default size and position.

7. Potentiometric Surface on Cross Section (Figure 164). This group appears only if the result data type is Hydraulic Head. See Section 4.2.2.2 for details.

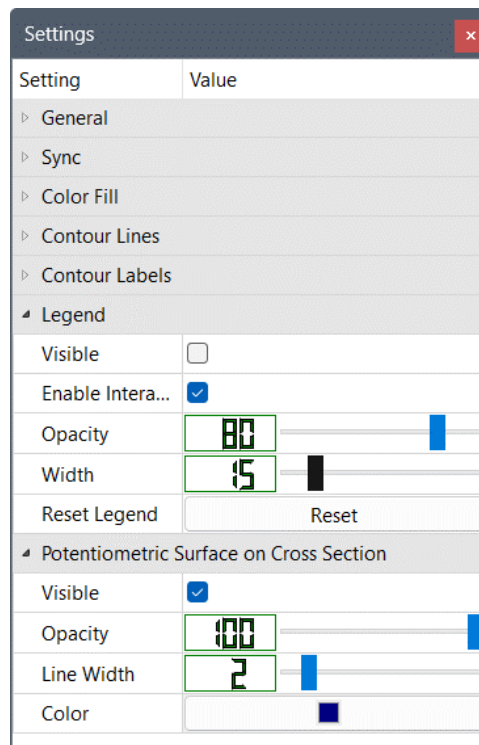


Figure 164. The Legend and Potentiometric Surface on Cross Section groups of the Settings window for Contour Maps

4.2.2. Result Contours

4.2.2.1. Create Contour Maps Based on Model Results

You can create a contour map based on the model result data by right-clicking on Result Contours on the TOC, select *Add* from a popup menu, and then select a data type from the Select Data dialog box. Contours will not be created for the areas of no-flow or dry cells, or areas with the reference value (see Reference in Section 4.2.1.2 above) equals to -1E20. In addition, the areas of inactive concentration cells will be omitted for concentration contours.

Contour maps can be created based on the following result types:

1. Hydraulic head: the default output file “modflow.head” in the model folder will be used.
2. Drawdown: the default output file “modflow.drawdown” in the model folder will be used.
3. Concentration: You will need to select a species and select either the dissolved or sorbed phase from the Select Data dialog box. The name of a result file is MT3Dnnn.UCN for the dissolved phase and MT3Dnnns.UCN for the sorbed phase, where nnn is the species number.

4. **Cell-By-Cell Flow Terms:** the default output file “modflow.flow” in the model folder will be used. The following supported data types are stored in that file: Storage, Interbed Storage, Constant Head, Drains, Drains with Return Flow, General-Head Boundary, River Leakage, Stream Leakage, Stream Flow Out, Evapotranspiration, Evapotranspiration Segments, Recharge, Flow Right Face, Flow Front Face, Flow Lower Face, Horizontal Flux (flux at the cell center based on fluxes across the horizontal cell faces), Vertical flux (flux at the cell center based on fluxes across the top and bottom cell faces), Flux (flux at the cell center based on fluxes across all cell faces). The flow values are given in volume per time unit per area [$L^3/(TL^2)$] or [L/T] for Evapotranspiration, Recharge, River/Stream leakage, Drain, and General-Head Boundary; the other flow values are expressed in volume per time unit [L^3/T].
5. **Interbed Storage Package (IBS):** The following data types are supported:
 - **Compaction:** The compaction values of each model layer with interbed storage are stored in the “modflow.ibscompaction” file.
 - **Subsidence:** Land surface subsidence is calculated as the sum of the compaction of all model layers. The calculated subsidence values are assigned to model layer 1 and stored in the “modflow.ibssubsidence” file.
 - **Critical Head:** The critical head (pre-consolidation head) of each model layer with interbed storage are stored in the “modflow.ibscriticalhead” file.
6. **Subsidence and Aquifer-System Compaction Package (SUB):** The following data types are supported:
 - **Compaction:** The compaction values of each model layer are stored in the “modflow.compaction” file.
 - **Subsidence:** Land surface subsidence is calculated as the sum of the compaction of all model layers. The calculated subsidence values are assigned to model layer 1 and stored in “modflow.subsidence” file.
 - **Vertical displacement:** The vertical displacement values of each model layer are stored in the “modflow.zdisplacement” file.
 - **Critical Head of no-delay interbeds:** The calculated critical head (pre-consolidation head) of no-delay interbed systems are stored in the “modflow.nodelay_hc” file. If a model layer has multiple no-delay interbed systems, the critical head of the first no-delay interbed system will be used for contouring.
 - **Critical Head of delay interbeds:** The calculated critical head (pre-consolidation head) of delay interbed systems are stored in the “modflow.delay_hc” file. If a model layer has multiple delay interbed systems, the critical head of the first delay interbed system will be used for contouring.
7. **Seawater Intrusion 2 Package:**
 - The calculated interface elevation (Zeta surface elevation) are stored in the output file “modflow.swi2zeta”. On the Map Viewer, the calculated zeta surface elevation may be used to create contour maps. On the cross-section window, the position of the zeta surface is marked with open circles (see Figure 165 for an example.) The display



Figure 165. The position of zeta surface is marked with open circles on the cross section

settings of the markers can be adjusted with the controls in *Zeta Marker on Cross Section* group of the Settings window.

- The other supported data types are stored in the “modflow.flow” file: Flow Right Face Correction, Flow Lower Face Correction, Flow Front Face Correction, and Constant Head Flow Correction. These correction terms are intended to be added to the corresponding cell-by-cell flow terms to yield the final flow results.

After adding a Contour Map item to the TOC, you can right-click on the item to access the following functions:

1. Show All Items: the contour map contains several items, one for each contour level. Select this to display all contour levels.
2. Hide All Items: Select this to hide all contour levels.
3. Export to Shapefile: this will display the Save As dialog box where you can select a shapefile (with the extension .shp) for storing the contour map.
4. Export Matrix Data: Select this to open the Export Data dialog box that allows you to export the matrix data of the displayed results. See Figure 48 on page 85 for details about that dialog box.
5. Export Groundwater Table: if the contour map is based on the calculated hydraulic head, this item allows you to export the head values of highest active cells (in each vertical columns of model cells) to a 2D Matrix file. If all model cells in a vertical column are dry or no-flow, the HDRY value is exported for that vertical column. The value of HDRY is defined in Models > Modflow > Output Control (Section 3.9.1.3).
6. Set as Starting Head: if the contour map is based on the calculated hydraulic head, this item allows you to set the calculated hydraulic head values (of all layers) as the starting hydraulic head values. For dry cells, the starting hydraulic head values are set to the cell bottom elevations. This feature is useful if you plan to run a transient flow model with the starting hydraulic head values based on the results of a prior model.
7. Copy Current Layer: Select this to open the Destination dialog box that allows you to copy the matrix data of the current layer to the corresponding layer of the selected destination. If the selected destination is a Transient Sandbox, the matrix data will be saved to the current stress period of the selected Transient Sandbox.
8. Copy All Layers: Select this to open the Destination dialog box that allows you to copy the matrix data of all layers to the selected destination. If the selected destination is a

Transient Sandbox, the matrix data will be saved to the current stress period of the selected Transient Sandbox.

9. Remove Item: Select this to remove the Contour Map item.

4.2.2.2. Customize Contour Maps

Most of the visualization settings of a result contour map are the same as those described in Section 4.2.1.2. The following is a list containing the result-contour specific settings:

1. General

- **Result File** is the (read-only) path to the binary file that contains the cell-by-cell model result values. Refer to Section 4.2.1.1 for the output file names of the supported models.

2. Sync

- **Mode**
 - **Auto:** The controls of the Sync group are deactivated and synced with the corresponding controls on the toolbar. Whenever the model is run, or any of the corresponding controls on the toolbar is changed, the contour map will be updated with the most recent model results.
 - **Lock:** The stress period, layer, date/time, and output times controls are deactivated and locked at their current settings. The contour map will never change, even if the model results have changed.
 - **Manual:** The stress period, layer, date/time, and output times controls are activated when applicable. The contour map will be based on the model results of the date/time defined by the Date/Time control.
- **Date/Time** is the date/time for which the model result is displayed.
 - If the selected Mode is Auto, Date/Time will be set to the end of the stress period and time step defined on the toolbar.
 - If the selected Mode is Manual, you may edit the date/time. After editing of the date/time is finished⁸², the values for Stress Period and Time Step will update accordingly and the contour map will be updated with “model result – reference” at the given date/time.
- **Output Times:** This is available for concentration contours. The dropdown box contains the output times that are found in the selected concentration file. When an item of the Output Times dropdown box is selected, the value for the Date/Time control will be set to the selected item and the contour map will be updated with “model result – reference” at the given date/time.
- **Stress Period and Time Step:** When the stress period or time step is changed, the value for the Date/Time control will be set to the end of the given stress period and

⁸² Editing is finished when the Date/Time control loses focus or when enter is pressed.

- time step. The contour map will be updated with “model result – reference” at the given date/time.
- **Layer:** This is the model layer for which the “model result – reference” is displayed on the Map Viewer. This control does not affect the display on the Row Cross Section tool.
- 3. Potentiometric Surface on Cross Section:** This group appears only if the result data type is Hydraulic Head. The potentiometric surface on a vertical column of model cells is defined as the hydraulic head at the highest active cell. The potentiometric surface on a cross section is formed by linking the potentiometric surface of individual vertical columns. The following settings define the appearance of the potentiometric surface on the model cross-section.
- **Visible:** Check this box to display the potentiometric surface on the cross-section window.
 - **Opacity:** The potentiometric surface is opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
 - **Line Width:** This is the line width (in pixels) that is used to draw the potentiometric surface.
 - **Color:** Use this dropdown box to select a color for the potentiometric surface.
- 4. Zeta Marker on Cross Section:** This group appears only if the result data type is Zeta Surface. The settings define the appearance of the markers that mark the position of the zeta surface on the model cross-section.
- **Visible:** Check this box to display the zeta markers on the cross-section window.
 - **Opacity:** The markers are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
 - **Line Width:** This is the line width (in pixels) that is used to draw the markers.
 - **Size:** This defines size of the zeta markers.
 - **Color:** Use this dropdown box to select a color for the zeta markers.

4.2.3. Flow Vectors

You can create flow vectors for individual cells by right-clicking on this item and then select *Add* from a popup menu. Flow vectors are based on the computed cell-by-cell flow terms⁸³, head values, effective porosity, and the geometry of the cells. The following input parameters are supported:

⁸³ Flow correction terms from the SWI2 package are included in the calculation and display of flow vectors.

1. General (Figure 166)

- **Name:** The text string is the name of the Flow Vectors Map displayed on the TOC.
- **Notes:** The text string can be used for notes.
- **Display Layer:** This is the layer number of the contour map. When multiple items are displayed, the items with smaller layer numbers are displayed on top of the items with larger layer numbers.
- **Popup on Hover:** If this is checked, when the mouse cursor hovers over a vector, a popup window containing detailed data of the vector will be displayed.

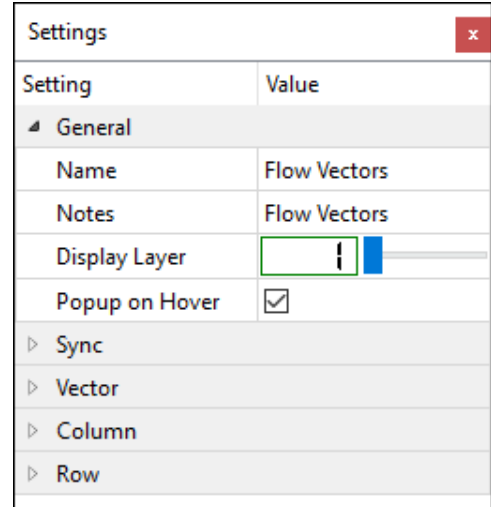


Figure 166. The General group of the Settings window for Flow Vectors

2. Sync (Figure 167)

- **Mode**
 - **Auto:** The stress period, time step and layer controls are deactivated and synced with the corresponding controls on the toolbar. Whenever the model is run or any of the controls is changed, the contour map will be updated based on the most recent model results.
 - **Lock:** The stress period, layer, species controls are deactivated and locked at their current settings. The contour map will never change, even if the model results have changed.
 - **Manual:** The stress period, layer, species controls are activated when applicable. The contour map will be created based on the model results of the specified stress period, time step, and species when applicable.
- **Date/Time** is the date/time for which the model result is displayed. After editing of the date/time is finished, the values for Stress Period and Time Step will update accordingly and the flow vectors will be updated with the model result at the given date/time.
- **Stress Period** and **Time Step:** When the stress period or time step is changed, the value for the Date/Time control will be set to the end of the given stress period and time step. The flow vectors will be updated with model result at the given date/time.
- **Layer:** This is the model layer for which the flow vectors are displayed.

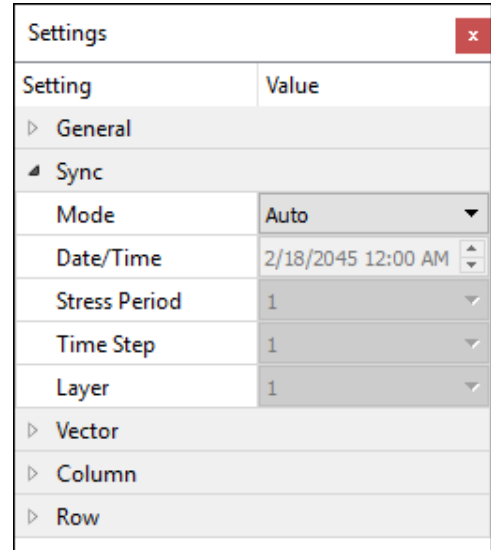


Figure 167. The Sync group of the Settings window for Flow Vectors

3. Vector (Figure 168)

- **Opacity:** The flow vectors are if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Line Width** is the line width (in pixels) that is used to draw the flow vectors.
- **Color:** Use this dropdown box to select a color for the flow vectors.
- **Data Type:**
 - **Volumetric Flow Rate [L³T⁻¹]:** The flow vectors are scaled by groundwater flow rate.
 - **Volumetric Flow Rate (Log) [L³T⁻¹]:** The flow vectors are scaled by logarithmic value of groundwater flow rate.
 - **Darcy Velocity [LT⁻¹]:** The flow vectors are scaled by the Darcy velocity.
 - **Darcy Velocity (Log) [LT⁻¹]:** The flow vectors are scaled by logarithmic value of the Darcy velocity of groundwater.
 - **Pore Velocity [LT⁻¹]:** The flow vectors are scaled by the pore velocity which is equal to Darcy velocity divided by the effective porosity.
 - **Pore Velocity (Log) [LT⁻¹]:** The flow vectors are scaled by logarithmic value of the pore velocity.
- **Vector Size** is a scaling factor of the display length of flow vectors. The display length of a vector is determined based on its magnitude of the selected Data Type and Upper Threshold (see below).
- **Lower Threshold** and **Upper Threshold:** A flow vector is displayed only if its magnitude is between the lower threshold (inclusive) and the upper threshold (inclusive). Assign zero or a negative value to deactivate a threshold. The default values of the Lower and Upper thresholds are the respective minimum and maximum values of pore velocity of the current time step and stress period.
 - If Upper Threshold is equal to or less than zero, the display length of a vector is scaled between 0 and the highest magnitude of the selected *Data Type* in the current layer. That is, the flow vectors of different model layers or simulation times may be scaled differently.
 - If Upper Threshold is greater than zero, the display length of a vector is scaled between 0 and Upper Threshold. If you need to compare flow vectors of different model layers or simulation times, assign a positive value to Upper Threshold (and a positive value to Lower Threshold, if the selected Data Type involves log-transformed values).

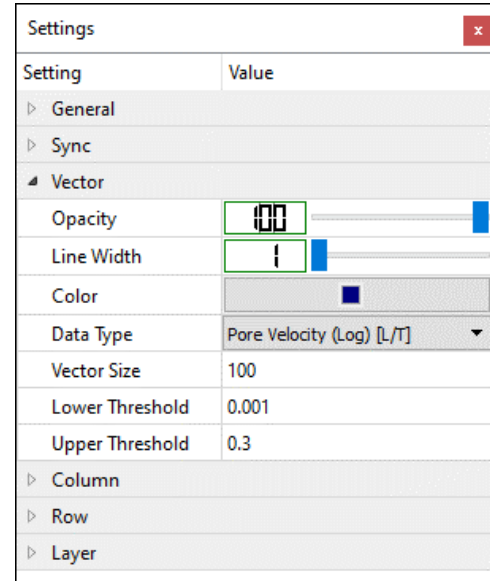


Figure 168. The Vector group of the Settings window for Flow Vectors

4. **Column, Row, and Layer (Figure 169):** The values of these groups define the region of interest. Only the flow vectors that lie within the region of interest are displayed. The Interval values specify how often the vectors should be displayed. For example, if start row is 5, end row is 16, and the row interval is 3, then the vectors at the rows 5, 8, 11, and 14 are displayed. The values of the Layer group only apply to the vectors on the Row Cross Section Window or Column Cross Section Window. The values of the Column and Row groups only apply to the vectors on the Map Viewer.

Settings x	
Setting	Value
▷ General	
▷ Sync	
▷ Vector	
▾ Column	
Start	1 ▼
End	60 ▼
Interval	1 ▼
▾ Row	
Start	1 ▼
End	1 ▼
Interval	1 ▼
▾ Layer	
Start	1 ▼
End	1 ▼
Interval	1 ▼

Figure 169. The Column and Row groups of the Settings window for Flow Vectors

4.2.4. Particle Tracking with MODPATH⁸⁴

4.2.4.1. Create MODPATH Instances

You can create a MODPATH instance by right-clicking on *MODPATH* on the TOC and then selecting *Add* from a popup menu to add a *Modpath Instance* to the MODPATH group. You can right-click on the MODPATH instance to access the following functions:

1. **Remove Item:** This will remove the MODPATH instance.
2. **Delete All Starting Points:** Delete all particles (starting points).
3. **Import Starting Points:** This will display the Import Starting Points dialog box (Figure 170) that assists you to import starting points (particles). This dialog box has the following features.

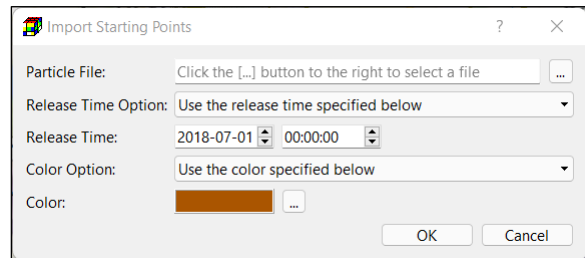



Figure 170. The Import Particles dialog box

- **Particle File:** You can import particles from particle files to MODPATH instances. Click on the  button to select a Particle file. See Section 5.1.7 for the Particle file format.
 - **Release Time Option:** This drawdown box has two options. The first option allows you to use the *simulationTime* stored in the particle file as the particle release time. The second option allows you to set the release time defined in this dialog box to the imported particles.
 - **Release Time:** This is the particle release date/time that will be assigned to the imported particles if the Release Time Option is “Use the release time specified below”.
 - **Color Option:** This drawdown box has two options. The first option assigns the colors stored in the particle file to the imported particles. The second option allows you assign the color defined in this dialog box to the imported particles.
 - **Color:** This is the color that will be assigned to the imported particles if the Color Option is “Use the color specified below”.
4. **Export Starting Points:** this will display the Save As dialog box where you can select a file for exporting starting points (particles) data to. See Section 5.1.7 for the Particle file format.
 5. **Export End Points:** this will display the Save As dialog box where you can select a file for exporting end points data to. See Section 5.1.7 for the Particle file format.
 6. **Export to Shapefile:** this will display the Save As dialog box where you can select a ZIP file for storing the exported shapefiles including pathlines, starting points, end points, and time marks.

⁸⁴ MODPATH does not account for the cell-by-cell flow correction terms of SWI2.

4.2.4.2. Time Concepts of MODPATH


MODPATH uses the following time concepts. The terms defined below are expressed in the model time unit.


1. Simulation Time is the elapsed time since the start of the MODFLOW simulation. Simulation time starts at 0 and increases throughout the course of a MODFLOW simulation.
2. Reference Time is a specific point in the Simulation Time that marks the beginning of a MODPATH particle tracking simulation.
3. Tracking Time: This is the elapsed time relative to Reference Time. Tracking time is 0 at Reference Time. Tracking time measures the accumulated time that a particle traveled during a particle-tracking analysis. The value of tracking time is always positive, regardless of whether particles are tracked in the forward or backward direction. For example, if the reference time is 500, a Tracking Time of 200 equivalents to the Simulation Times of 700 and 300 for forward and backward particle tracking simulations, respectively.

4.2.4.3. Add Particles

In MODPATH, particles (starting points) may be released at different time points. The release time of a particle is defined based on the settings when the particle is added to the map. For forward tracking, the release time of new particles is set to the beginning of the current time step and stress period. For backward tracking, the release time is set to the end of the current time step and stress period. Once particles are added to the map, their release times, start locations, and display colors are fixed.

There are three ways to add particles.

1. Import from a Particle File
 - Right-click the MODPATH instance, select Import Starting Points, then use the Import Particles dialog box to import particles. See Section 4.2.4.1 for details.
2. Add Individual Particles (ctrl+click)
 - Click the MODPATH instance to make it the active layer and the click on the MODPATH Tool button  on the toolbar.
 - Modify the *Tracking Direction* and the settings of the *Particle Start Point* group of the MODPATH instance as needed. See Section 4.2.4.5 for details.
 - Move to the layer, where the particles should be placed.
 - Ctrl+click the mouse at the desire location on the map to add a particle. The vertical coordinate of the particle is defined by $z = z_{bot} + z_L \cdot (z_{top} - z_{bot})$, where z_{bot} and z_{top} are the elevations of layer bottom and top, respectively, and z_L is the local vertical coordinate defined by Local Z (see Figure 174) in the settings window.
3. Add a Group of Particles (ctrl+drag)

- Click the MODPATH instance to make it the active layer and then click on the MODPATH Tool button  on the toolbar.
- Modify the *Tracking Direction* and the settings of the *Particle Start Point* group of the MODPATH instance as needed. See Section 4.2.4.5 for details.
- Move to the layer, where the particles should be placed.
- Ctrl+drag a window covering the region where particles should be placed, then release the mouse button to open the Add Particles dialog box (Figure 171). Use the dialog box to add particles. The features of the dialog box are as follows.

Use the dialog box to add particles. The features of the dialog box are as follows.

- **Area of Interest:** particles will be placed on faces of the cells, within the cells, or on circles centered at the cells that are located inside the selected area of interest

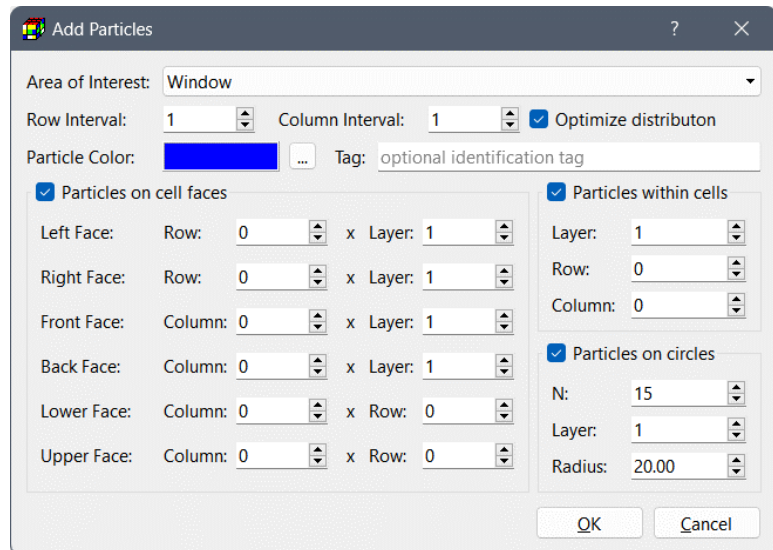



Figure 171. The Add Particles dialog box



from this dropdown box, which contains the following items:

- Window: this item represents the drawn window.
- Views: all views (see Section 2.1) that are created under Views in the TOC can be used as an area of interest. Using a view, you can place particles to cells that are located within an irregular polygon.
- **Row Interval and Column Interval:** Particles are assigned to cells that are spaced by given row and column intervals and that are located within the selected area of interest.
- **Optimize Distribution:** Check this box to ensure that particles are even distributed over cells when placing particles on cell faces.
- **Particle Color** is the color for new particles. To change a color, click on the  button and then pick a color from the Select Color dialog box. Color is included in the exported shapefiles of the particle tracking results.
- **Tag:** This is an optional text string that can be used to identify particles and their associated pathlines/time marks. Tag is limited to 50 characters and comma and space are not allowed. Tag is included in the exported shapefiles of the particle tracking results.
- **Particles on cell faces:** Check the box to activate this group. It controls the number of particles that will be placed on the faces of the cells that are located within the

selected Area of Interest. The number of particles on each face is equal to the product of the respective specified numbers.

- **Particles within cells:** Check the box to activate this group. It controls the number of particles that will be placed within the cells that are located inside the drawn box. The number of particles for each cell is equal to the product of the specified values for *Layer*, *Row*, and *Column*.
- **Particles on circles:** Check the box to activate this group. It controls the number of particles that will be placed on circles centered at the cells that are located inside the drawn box. *N* is the number of particles on a circle, *Layer* is the number of circles, and *Radius* is the circle radius.

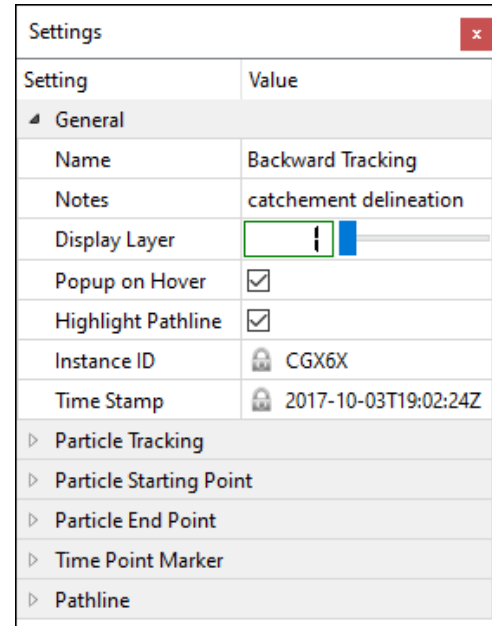
4.2.4.4. Delete Particles

4. Delete Particles of a Layer (shift+click or shift+drag)
 - Click the MODPATH instance to make it the active layer and the click on the MODPATH Tool button  on the toolbar.
 - Move to the layer, where the particles should be deleted.
 - Shift+click on a particle to delete it, or shift+drag a window encompassing the particles that should be deleted, and then release the mouse button.
5. Delete Particles of all Layers (alt+click or alt+drag)
 - Click the MODPATH instance to make it the active layer and the click on the MODPATH Tool button  on the toolbar.
 - Press and hold the Control-key and the left mouse button, move the mouse to drag a box encompassing the particles that should be deleted, and then release the Control-key and left mouse button.
6. Delete All Particles
 - Right-click the MODPATH instance and select *Delete All Starting Points*.

4.2.4.5. Customize MODPATH Settings

1. General (Figure 172)

- **Name:** The text string is the name of the MODPATH instance displayed on the TOC.
- **Notes:** The text string can be used for notes.
- **Display Layer:** This is the layer number of the particles, time marks, and pathlines. When multiple items are displayed, the items with smaller layer numbers are displayed on top of the items with larger layer numbers.
- **Popup on Hover:** If this is checked and when the mouse cursor hovers over an object, such as a particle, time mark, or pathline, a popup window containing detailed data of the object will be displayed and the corresponding pathline will be highlighted.
- **Highlight Pathline:** If this is checked and when the mouse cursor hovers over an object, such as a particle or time mark, highlight the pathline containing that object.
- **Instance ID:** This is the unique identifier of the MODPATH instance. The base filename of the input and output files of this MODPATH instance is MODPATH-InstanceID.
- **Time Stamp:** This is the time stamp of the last MODPATH run.



The screenshot shows a 'Settings' window with a 'General' group expanded. The settings are as follows:

Setting	Value
General	
Name	Backward Tracking
Notes	catchement delineation
Display Layer	1
Popup on Hover	<input checked="" type="checkbox"/>
Highlight Pathline	<input checked="" type="checkbox"/>
Instance ID	CGX6X
Time Stamp	2017-10-03T19:02:24Z
Particle Tracking	
Particle Starting Point	
Particle End Point	
Time Point Marker	
Pathline	

Figure 172. The General group of the Settings window for MODPATH

2. Particle Tracking (Figure 173)

- **MODPATH:** Click the *Run* button to build input files to MODPATH and then start MODPATH. The run progress will be displayed in the MODPATH Output window. You may terminate the run at any time by clicking on the Terminate button. After completing the MODPATH run, the pathlines and time point markers will be displayed automatically.
- **Number of Time Points** is the number of time points at which the locations of particles should be saved in an output file.
- **Time Point Interval** is the interval of time (expressed in the model time unit) between time points. A Time Point Marker is placed whenever the *Tracking Time* is a multiple of the given Time Point Interval.

- **Tracking Direction** defines whether the simulation tracks particles forward or backward in time. The tracking direction affects the release time of particles.
 - **Forward:** When particles are added to the map, their release time is the beginning of the current stress period/time step.
 - **Backward:** When particles are added to the model, their release time is the end of the current stress period/time step. Thus, if you want to simulate backward particle tracking from the end of the flow simulation, you need to choose this setting and set the stress period/time step to the last ones, before adding new particles.
- **Reference Time** is a specified point of Simulation Time that marks the beginning of a MODPATH particle tracking simulation. If the **Auto** box is checked, *Reference Time* is to 0 (start of the flow simulation) for forward particle tracking and is set to *Simulation Length* (end of the flow simulation) for backward particle tracking.

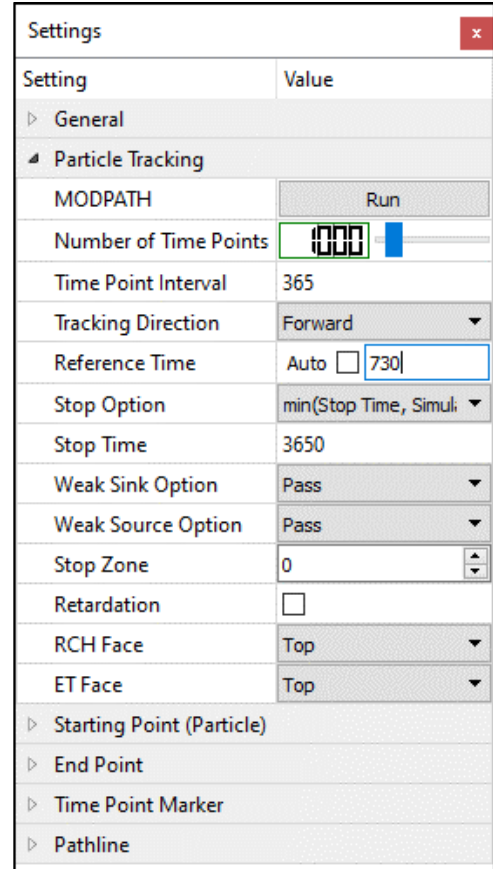


Figure 173. The Particle Tracking group of the Settings window for MODPATH

- **Stop Option:**
 - **Simulation Limit:**
 - For forward tracking, stop the particle tracking simulation at the end the final time step of the MODFLOW simulation.
 - For backward tracking, stop the particle tracking simulation at the beginning of the MODFLOW simulation.
 - **Extended:**
 - For forward tracking, extend the final steady-state time step (if applicable) and track all particles until they reach termination locations.
 - For backward tracking, extend the initial steady-state time step (if applicable) and track all particles until they reach termination locations.
 - **min(Stop Time, Simulation Limit):** Particles are stopped when the *Stop Time* (see below) or Simulation Limit is reached.
- **Stop Time** is the *Tracking Time* at which a particle tracking simulation is stopped. Stop Time is used only if the selected Stop Option is *min(Stop Time, Simulation Limit)*.

Simply put, a forward particle tracking simulation stops at the simulation time equal to $[Reference\ Time + Stop\ Time]$. A backward particle tracking simulation stops at the

simulation time equal to $[Reference\ Time - Stop\ Time]$. For example, assuming you have a flow model with the simulation length of 1000. Setting *Stop Time* to 300 and *Reference Time* to 100 will stop a forward particle tracking at the simulation time 100 + 300. Setting *Stop Time* to 300 and *Reference Time* to 900 will stop a backward particle tracking at the simulation time 900 – 300.

- **Weak Sink Option** defines whether particles should be stopped when they enter weak sink cells or should pass through weak sink cells. A strong sink cell is one that contains an internal sink and has inflow from adjacent cells but no outflow to other cells. In contrast, weak sink cells have both inflow and outflow across their cell faces.
- **Weak Source Option** defines whether particles should be stopped when they enter weak source cells or should pass through weak source cells. A strong source cell is one that contains an internal source and has outflow to adjacent cells but no inflow from other cells. In contrast, weak source cells have both inflow and outflow across their cell faces.
- **Stop Zone:** A value greater than zero activates the Stop Zone option of MODPATH. A value of zero deactivate the Stop Zone option. When the Stop Zone option is activated, particles will stop when entering cells with the same stop zone value as given here. Use *Parameters > Stop Zone* to assign stop zone values to cells.
- **Retardation:** The travel velocity of a particle is set to its pore velocity divided by Retardation. Particles move slower with a larger Retardation value. The retardation values are defined by *Parameters > Retardation Factor* with the Data Editor.
- **RCH Face:** recharge may be treated as a distributed source or assigned to a cell face by selecting a corresponding option.
- **ET Face:** evapotranspiration may be treated as a distributed sink or assigned to a cell face by selecting a corresponding option.

3. Particle Starting Point (Figure 174)

- **Opacity:** The particle starting points are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
- **Size:** This defines the display size of all particle starting points.
- **Color:** Use this dropdown box to select a color for new particles that are added to the map individually as described in Section 4.2.4.3.

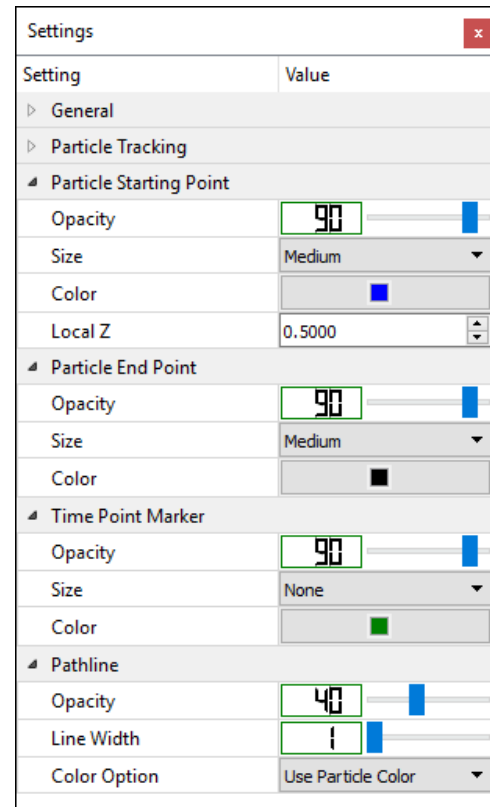


Figure 174. The Particle Starting Point, Particle End Point, Time Point Marker, and Pathline groups of the Settings window for MODPATH

- **Local Z:** This is the local z coordinate of new particles that are added to the map individually as described in Section 4.2.4.3.
- 4. Particle End Point (Figure 174):** particle end points are displayed as diamonds on the map viewer, and they represent the particle stop locations.
- **Opacity:** The particle end points are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
 - **Size:** This defines the display size of the particle end points.
 - **Color:** Use this dropdown box to select a color for the particle end points.
- 5. Time Point Marker (Figure 174)**
- **Opacity:** The time point markers are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
 - **Size:** This defines the display size of the time point markers.
 - **Color:** Use this dropdown box to select a color for the time point markers.
- 6. Pathline (Figure 174)**
- **Opacity:** The pathlines are opaque if the opacity value is 100, translucent if the opacity is between 1 and 99, and transparent if the opacity is 0.
 - **Line Width** is the line width (in pixels) that is used to draw the pathlines.
 - **Color Option**
 - Use Particle Color: Pathlines are displayed in the same color as the respective particle starting points.
 - Use Layer Color: Pathlines are displayed in the color of the corresponding layer where the pathlines are located. The layer colors are defined in the Layer Property dialog box. See Section 3.7.3 for details.

This Page Intentionally Left Blank

Chapter 5. Supplementary Information

5.1. File Formats

5.1.1. 2D Matrix and 2D Weight Matrix

A 2D Matrix or 2D Weight Matrix file is a text file with the following format that can be exported from or imported to the Data Editor as described in Section 3.6.20. The first two lines of a 2D Matrix file can be omitted if it has the same dimension as the model

File Format

1. #2DMATRIX
2. NCOL, NROW
3. MATRIX

Description

- **#2DMATRIX:** The first line holds the text string “#2DMATRIX” that serves as the file format identifier.
- **NCOL, NROW** are, respectively, the numbers of columns and rows of the matrix that follows.
- **MATRIX** is a two-dimensional data matrix saved row by row. The data of the first model row is given on the first row of the file, the second follows, and so on. Each row contains NCOL values from left to the right. The values must be separated by a space or comma.

Example

The following is an example showing a 2D Matrix file for a grid with 4 columns and 3 rows. The grid values are representing by a 4 x 3 matrix.

```
#2DMATRIX
4, 3
11, 12, 34, 23
42, 35, 23, 21
23, 41, 31, 19
```

5.1.2. 2D Modflow Array

A 2D Modflow Array file is a text file with the following format that can be exported from or imported to the Data Editor as described in Section 3.6.20.

File Format

1. #2DMODFLOWARRAY
2. NCOL, NROW
3. ARRAY

Description

- **#2DMODFLOWARRAY:** The first line holds the text string “#2DMODFLOWARRAY” that serves as the file format identifier.
- **NCOL, NOW** are, respectively, the numbers of columns, and rows of the array that follows.
- **ARRAY** is a two-dimensional data array saved in the format as described in the section “Input Instructions for Array Reading Utility Subroutines” of Chapter 8 in (Harbaugh, 2005). Note that arrays saved in external binary files are not supported.

Example

The following is an example showing a 2D Modflow Array file for a grid with 4 columns and 3 rows. The grid values are representing by a 4 x 3 matrix.

```
#2DMODFLOWARRAY
4, 3
INTERNAL 1 (FREE) -1
11, 12, 34, 23
42, 35, 23, 21
23, 41, 31, 19
```

The following is an example showing a 2D Modflow Array file for a grid with 9 columns and 7 rows. The grid has a constant value of 526.

```
#2DMODFLOWARRAY
9, 7
Constant 526
```

5.1.3. 3D Modflow Array

A 3D Modflow Array file is a text file with the following format that can be exported from or imported to the Data Editor as described in Section 3.6.20.

File Format

1. #3DMODFLOWARRAY
2. NCOL, NROW, NLAY
3. (the following array repeats NLAY times)
4. ARRAY

Description

- **#3DMODFLOWARRAY:** The first line holds the text string “#3DMODFLOWARRAY” that serves as the file format identifier.
- **NCOL, NROW** are, respectively, the numbers of columns and rows of the arrays that follow this line.
- **NLAY** is the number of layers.
- **ARRAY** is a two-dimensional data array saved in the format as described in the section “Input Instructions for Array Reading Utility Subroutines” of Chapter 8 in (Harbaugh,

2005). Note that arrays saved in external binary files are not supported. ARRAY is repeated NLAY times. The first occurrence of the array is for the first (top) model layer, the second occurrence for the second layer, and so on.

Example

The following is an example showing a 3D Modflow Array file for a grid with 4 columns, 3 rows, and 2 layers. For the first layer, the grid values are representing by a 4 x 3 matrix. For the second layer, the grid has a constant value of 37.

```
#3DMODFLOWARRAY
4, 3, 2
INTERNAL 1 (FREE) -1
11, 12, 34, 23
42, 35, 23, 21
23, 41, 31, 19
Constant 37
```

5.1.4. Comma-Separated Values File

A Comma-Separated Values file is a text file with the following format that can be exported from or imported to the Data Editor as described in Section 3.6.20.

File Format

1. #CSV
2. X, Y, Z (repeat this line as many times as needed)

Description

- **#CSV:** The first line holds the text string “#CSV” that serves as the file format identifier.
- **X, Y, Z:** Each line represents a data point, where [x, y] are the coordinates of the point, and z is the value at the point.
 - If the Geographic coordinate system is used: If an optional prj file⁷ exists in the same folder as the CSV file, [x, y] are expressed in the coordinate system defined by the prj file. If a prj file does not exist, [x, y] are assumed to be [longitude, latitude], where longitude and latitude are expressed in WGS84.
 - If the Cartesian coordinate system is used: [x, y] are expressed in the Cartesian coordinate system.

5.1.5. ASCII Raster

ASCII Raster, also known as ASC/INFO ASCII GRID⁸⁵ is a non-proprietary raster GIS file format developed by ESRI. ASCII Raster files may be exported or imported via the Data Editor, see

⁸⁵ For details, see <https://www.loc.gov/preservation/digital/formats/fdd/fdd000421.shtml>

Section 3.6.20 for details. ASCII Raster files may also be created by GIS applications, such as ArcMap or QGIS.

File Format

The first six lines contain metadata of the grid. Each line contains a label and a value. These lines are followed by a matrix with values listed in the order they would naturally appear (left-right, top-down).

```
ncols      xxx
nrows      xxx
xllcorner  xxx
yllcorner  xxx
cellsize   xxx
NODATA_value xxx
ZM1 ZM2 ZM3 ... ZMn
...
Z31 Z32 Z33 ... Z3n
Z21 Z22 Z23 ... Z2n
Z11 Z12 Z13 ... Z1n
```

Description

1. `ncols` and `nrows` are the numbers of columns and rows, respectively.
2. `xllcorner` and `yllcorner`⁸⁶ are, respectively, the western (left) x-coordinate and southern (bottom) y-coordinate of the lower-left corner of the grid. Instead of `xllcorner` and `yllcorner`, `xllcenter` and `yllcenter` may be used. `xllcenter` and `yllcenter` are, respectively, the x and y coordinates of the center of the lower-left cell of the grid.
3. `cellsize`⁸⁶ is the length of one side of a square cell. Instead of “`cellsize xxx`”, a line with “`DX xxx`” and a line with “`DY xxx`” may be used. `DX` and `DY` are, respectively, the column and row widths of the raster.
4. `NODATA_value` is the value that is regarded as “missing” or “not applicable”.
5. The remainder of the file lists the raster values for each cell, starting at the upperleft corner. These values are delimited using a single space character.

Example

The following is an example showing an ASCII Raster file for a grid with 4 columns and 3 rows.

⁸⁶ If the Geographic coordinate system is used and a PRJ⁷ file containing the spatial reference information is available, `xllcorner`, `yllcorner`, and `cellsize` are expressed in the coordinate system defined in the PRJ file. If the Geographic coordinate system is used, but a PRJ⁷ file is not available, `xllcorner` and `yllcorner` are assumed to be expressed geographic longitude and latitude and `cellsize` is arc angle. If the Cartesian coordinate system is used, `xllcorner`, `yllcorner`, and `cellsize` are assumed to be expressed in the Cartesian coordinates, and an PRJ file is not used.


```
NCOLS 4
NROWS 3
XLLCORNER -117.73
YLLCORNER 33.451
DX 400
DY 350
NODATA_value -1E20
11 12 34 23
42 35 23 21
23 41 31 19
```

5.1.6. Shapefile

The shapefile format is a binary format that is described in detail in ESRI white paper (ESRI, 1998).

5.1.7. Particle File

A Particle file is a text file that stores the particle locations in the following format that can be exported from or imported to MODPATH instances. See Section 4.2.4 for details about MODPATH instances.

File Format

1. #PARTICLE or #STARTINGLOCATION
2. Version
3. N
4. (the following record repeats N times)
5. layer, row, column, localX, localY, localZ, releaseTime [, status, color, tag]

Description

- **#PARTICLE** or **#STARTINGLOCATION**: The first line holds the text string “#PARTICLE” or “#STARTINGLOCATION” that serves as the file format identifier.
- **Version** is reserved for file version. The default value of 1.0 should be used.
- **N** is the number of starting points (particles) saved in this file.
- Layer, row, column, localX, localY, localZ, simulationTime: each line represents a particle.
 - **layer**, **row**, and **column** are the cell where the particle is located.
 - **localX**, **localY**, and **localZ** are the local coordinates of the particle within the cell. The value of local coordinates range between 0 and 1.
 - **simulationTime** is the simulation time at which the particle is released.
 - **status** is an integer from one of the following the status categories. **Status** is optional but must be included if the optional **color** or **tag** (see below) is present.
 - 0 = Pending release
 - 1 = Active
 - 2 = Terminated at boundary face

- 3 = Terminated in weak sink cell
 - 4 = Terminated in weak source cell
 - 5 = Terminated in cell with no exit face
 - 6 = Terminated in cell with specified zone number
 - 7 = Terminated in an inactive cell
 - 8 = Permanently unreleased
 - 9 = Terminated for unknown reason
- o **color** is the particle color in the format "#RRGGBB"; i.e., a "#" character followed by three two-digit hexadecimal numbers. For example, #0056D6. **Color** is optional but must be included if the optional **tag** (see below) is present.
 - o **tag** is an optional text string that can be used in exported shapefiles for identifying particles and the associated pathlines/time marks.

5.1.8. Georeference file

A Georeference file is a text file with the following format. The file can be created by or loaded to the Georeference dialog box (see Figure 3).

File Format

1. #Georeference
2. Version
3. Method
4. Azimuth, Lat1, Lng1
5. LatUL, LngUL
6. LatUR, LngUR
7. LatLL, LngLL
8. LatLR, LngLR

Description

- **#GEOREFERENCE:** The first line holds the text string "#GEOREFERENCE" that serves as the file format identifier.
- **Version** is reserved for file version. The default value of 1.0 should be used.
- **METHOD** is the georeferencing method. Either 1 or 2 should be entered. If Method = 1, grid azimuth and coordinates of the lower-left grid corner, are used. If Method = 2, the coordinates of the four model grid corners are used to place the model grid on the map.
- **Azimuth, Lat1, Lng1:** These values are always read, but will be used only if Method = 1.
 - o Azimuth is the angle in degrees measured clockwise from a meridian to the axis along the model's row direction. Enter 90 if the grid is not rotated.
 - o Lat1 and Lng1: WGS-84 Latitude and longitude coordinates of the lower-left grid corner (i.e., outer corner of cell [NROW, 1]).

- **LatUL, LngUL, LatUR, LngUR, LatLL, LngLL, LatLR, LngLR:**
 - If the Geographic coordinate system is used, these values are, respectively, the WGS-84 Latitude and longitude coordinates of the upper-left (cell[1,1]), upper-right (cell[1, NCOL]), lower-left (cell[NROW, 1]), and lower-right (cell[NROW, NCOL]) corners of a model grid. These values are always read but will be used only if Method = 2.
 - If the Cartesian coordinate system is used, these values are, respectively, the Y and X Cartesian coordinates of the upper-left (cell[1,1]), upper-right (cell[1, NCOL]), lower-left (cell[NROW, 1]), and lower-right (cell[NROW, NCOL]) corners of a model grid. These values are always read but will be used only if Method = 2.

5.1.9. Observation File

Observation files are saved in the Microsoft Excel format. You can create a template file by right clicking on Observation Points on the TOC and then selecting Export to Excel File. See Section 3.6.3 for how to create or import an observation file. An observation file contains the following sheets in an arbitrary order. The sheet names are case-insensitive. The **observationpoints** sheet is required. All other sheets are optional.

- **observationpoints** – name, layer, and coordinates of observation points.
- **head** – hydraulic head or groundwater elevation
- **drawdown** – groundwater elevation drawdown
- **concentration** – groundwater concentration
- **compaction** – layer compaction
- **subsidence** – land surface subsidence
- **tmax** – maximum daily air temperature in the unit [temp_units] defined in GSFLOW > Settings.
- **tmin** – minimum daily air temperature in the unit [temp_units] defined in GSFLOW > Settings.
- **precip** – precipitation in the unit [precip_units] defined in GSFLOW > Settings.
- **humidity** – air humidity given as [decimal fraction], i.e., dimensionless, between 0 and 1
- **pan_evap** – pan evaporation in [inches]
- **potet** – potential ET in [inches] which may be used as the Climate by HRU input data to the precipitation distribution module of GSFLOW.
- **wind_speed** – wind speed in [meters per second] which may be used by the potet_pm_sta module for computing the potential evapotranspiration by using the Penman-Monteith formulation.
- **streamflow** – streamflow measurements in [L³/T] which may be used by Scatter Chart, Line Chart, and PEST.
- **snowdepth** – snow depth in [inches].
- **solrad** – solar radiation. The product of solrad and rad_conv is in [langleys]. The value rad_conv is defined in GSFLOW > Settings and is a factor to convert measured solar radiation to langleys.

- **stage** – stage measurements in [L] which may be used by Scatter Chart, Line Chart, and PEST.
- **layerweight** – layer weight factors of observation points. See Figure 39 and Equation (3) for the definition of layer weight. If both the LayerWeight and Screen sheets of an observation well are provided, the LayerWeight sheet takes priority over the Screen sheet.
- **screen** – screen intervals of observation points (wells).

The contents of the individual sheets are as follows.

1. **ObservationPoints:** The first row of this sheet contains the column headers. The remaining rows contain the data. The columns are as follows.
 - **Active** is either 1 (the observation point is active) or 0 (the observation point is inactive).
 - **PointName** is the unique name of the observation point.
 - **Label** will be displayed next to the observation point on the map. Use “\n” to move to next line; “\u” to move to previous line; and “\” for backslash. Only ASCII characters with ASCII values from 32 to 126 are supported.
 - **Layer** is the model layer, to which the observation point belongs. Set Layer to 0 for multi-layer observation points.
 - **Latitude/Y** and **Longitude/X:** If the Geographic coordinate system is used, **Latitude** and **Longitude** are the geographic coordinates of the observation point. If the Cartesian coordinate system is used, X and Y are the coordinates of the observation point.
 - **Elevation** is the ground surface elevation.
 - **XOffset** and **YOffset** are the easting and northing offsets (in meters) of the observation point label. These values can be used to adjust the display position of the label.
 - **Notes** is a text string.
 - **HeadTarget, DrawdownTarget, StageTarget, StreamflowTarget, ConcentrationTarget, CompactionTarget, SubsidenceTarget:** These columns contain 0, 1, or 2. These values defined how the observation data of an observation point are used.
 - A value of zero (0) means that observation data of the observation point will not be used in model calibration⁸⁷, time-series charts, or scatter charts.
 - A value of one (1) means that all observation data of the observation point will be used in model calibration⁸⁷, time-series charts, or scatter charts.
 - A value of two (2) means that only the selected observation data of the observation point will be used in model calibration⁸⁷, time-series charts, or scatter charts.

⁸⁷ The current version of PM does not support using concentration, compaction and subsidence as calibration targets with PEST.

2. **head, drawdown, tmax, tmin, precip, humidity, pan_evap, potet, wind_speed, streamflow, snowdepth, solrad, stage, compaction, and subsidence:** The first row of these sheets contains the column headers. The remaining rows contain the data. The columns are as follows.
 - **PointName** is the name of the observation point, at which the observations are made.
 - **SimulationTime** is the observation time, expressed in the simulation time. If you prefer to use calendar date/time, leave this cell blank.
 - **Value** is the observed value.
 - **Weight:** The Weight of an observation gives a relative confidence level of the observed value. The higher the Weight value, the better is the measurement. The weight can be set at zero if needed (meaning that the observation takes no part in the calculation of the objective function during a parameter estimation process), but it must not be negative. Refer to the PEST User Manual (Doherty, 2016) for the function of weights in the parameter estimation process.
 - **DateTime** is the observation calendar date/time, for example 12/31/2019 23:59:59⁸⁸. This value is used only if the SimulationTime cell is blank. During the import process, PM determines the corresponding SimulationTime based on the model start date/time and the given DateTime.
3. **Concentration:** The first row of this sheet contains the column headers. The remaining rows contain the data. The columns are as follows.
 - **PointName, Value, SimulationTime, Weight, and DateTime:** These are the same as the respective values described above for Head and Drawdown.
 - **Species:** Name of the species that is one of the Species Names defined in the Species tab of the Transport Settings dialog box (see Section 3.9.3.1).
4. **LayerWeight:** This sheet is used to assign layer weights of multi-layer observation wells. The first row of this sheet contains the column headers. The remaining rows contain the data. Repeat as many rows as needed for each observation well. Layer-weighted head values of multi-layer wells are used in Line or Scatter charts and used by PEST during parameter estimation process. See Section 3.6.3 for details. The columns are as follows.
 - **PointName** is the name of the observation point, at which the observations are made.
 - **Layer** is the model layer number.
 - **Weight** is the weight of the respective layer.
5. **Screen:** PM can calculate layer weights of multi-layer observation wells based on screen intervals provided in this sheet. The first row of this sheet contains the column headers. The remaining rows contain the data. Repeat as many rows as needed for each observation well. If both the **LayerWeight** and **Screen** sheets of an observation well are

⁸⁸ If the time portion is omitted, the time is assumed to be 00:00:00. Note that Excel internally stores the calendar date/time as a floating value. The displayed date/time string may be different based on your locale and/or format settings.

provided, the **LayerWeight** sheet takes priority over the **Screen** sheet. The columns are as follows.

- **PointName** is the name of the observation point.
- **Top** is the elevation of the top of the screen interval.
- **Bottom** is the elevation of the bottom of the screen interval.

5.1.10. PM Pilot Points File

A PM Pilot Points file is a comma-separated value (CSV) file with the following format⁸⁹ that can be exported from or imported to the Pilot Point Input Method of the Data Editor as described in Section 3.4.2.4.

File Format

1. #CSV PM_PILOTPPOINTS
2. #1.0
3. X, Y, PARVAL1[, PARLBND, PARUBND, PARTRANS, PARCHGLIM, PARGP, SCALE, OFFSET, TIEDDATATYPE, TIEDLAYER]

(each line represents a pilot point, repeat as many lines as needed. The values inside the brackets are optional. All of them can be omitted. However, if one of them is cited in a line, all preceding values in that line must be cited.)

Description

- **#CSV PM_PILOTPPOINTS:** The first line holds the text string that serves as the file format identifier.
- **#1.0:** this is the file version number. The value of the current version is 1.0.
- **X, Y** are the coordinates of the point.
 - If the Geographic coordinate system is used: If an optional prj file⁷ exists in the same folder as the Pilot Point file, [x, y] are expressed in the coordinate system defined by the prj file. If a prj file does not exist, [x, y] are assumed to be [longitude, latitude], where longitude and latitude are expressed in WGS84.
 - If the Cartesian coordinate system is used: [x, y] are expressed in the Cartesian coordinate system.
- **PARVAL1** is the parameter value at the pilot point.
- **PARLBND, PARUBND, PARGP, SCALE, and OFFSET:** These values are used in conjunction with PEST, see Section 3.9.5.2.1 for details.
- **PARTRANS:** This is an integer controlling the data transformation. The valid values are as follows: None = 0, Log-transformed = 1, Fixed = 2, and Tied = 3. See Section 3.9.5.2.1 for details about PARTRANS.
- **PARCHGLIM:** This is an integer used to designate whether the value change of a pilot point parameter is relative-limited (PARCHGLIM = 0) or factor-limited (PARCHGLIM = 1).

⁸⁹ Pilot Points files defined in Groundwater Data Utilities (Doherty, 2020) have a different format.

If the parameter is fixed (i.e., PARTRANS = 2), PARCHGLIM is not used but a value still must be provided.

- **TIEDDATATYPE:** This is either an integer or a string representing the data type that a pilot point is linked to. The valid case-insensitive strings are HK, VK, HANI, VANI, Ss, Sy, T, S, and VCONT. The corresponding integer values are as follows: HK = 1000, VK = 1001, HANI = 1003, VANI = 1004, Ss = 1005, Sy = 1006, T = 1009, S = 1010, VCONT = 1011.
- **TIEDLAYER:** This is the model layer where “parent” pilot points are located. A pilot point is tied to the nearest pilot point located in the TIEDLAYER model layer of the data type defined by TIEDDATATYPE.

5.1.11. Polygon File

A Polygon file is a text file with the following format. You can export or import polygon files via the Data Editor, see Section 3.4.2.3 for details.

File Format

1. #PM_POLYGONS
2. Version
3. NPolygons
4. (repeat the following data block, i.e., data 4 – 13, NPolygons times; a block for a polygon)
5. DataVersion
6. Name
7. IsOn, IsHole, AutoLayerNumber, ShowVertices, Priority
8. LineWidth, Red, Green, Blue
9. NP
10. Longitude/X, Latitude/Y (repeat NP times)
11. NValues
12. Value (an array of NValues values separated by space or in different lines)
13. NVars
14. Var (an array of NVars values separated by space or in different lines)
15. NConcs
16. Conc (an array of NConcs values separated by space or in different lines)

Description

- **#PM_POLYGONS:** The first line holds the text string “#PM_POLYGONS” that serves as the file format identifier.
- **Version** is the version number that is reserved for future use. The value for the current version is 1.
- **NPolygons** is the number of polygons.
- **DataVersion** is the version number of the data block. The value for the current DataVersion is 1.01
- **Name** is the name of the polygon

- **IsOn** is a value equals to either 1 or 0. The value of 1 means that the polygon is turned on; 0 means off.
- **IsHole** is a value equals to either 1 or 0. The value of 1 means that the polygon represents a hole; 0 means not a hole.
- **AutoLayerNumber** is a value equals to either 1 or 0. The value of 1 means that when applying the polygon to cells, the auto layer number feature is on; 0 means off.
- **ShowVertices** is a value equals to either 1 or 0. The value of 1 means that the vertices of the polygon is displayed; 0 means not displayed.
- **Priority**: When applying overlapping polygons to cells, the polygon with the smallest priority value has the highest priority.
- **LineWidth**: This value is reserved for future use. It should be 2.
- **Red, Blue, Green**: These values are reserved for future use. They should range between 0 and 255.
- **NP**: Number of vertices of the polygon. A polygon must have at least 6 vertices. The number of vertices must be even. The odd numbered vertices are big white vertices (see Section 3.4.2.3 for the vertex types), the even numbered vertices are small gray vertices.
- **Longitude/X, Latitude/Y** are coordinates of the vertex. If the Geographic coordinate system is used, they are expressed in the WGS 84 geographic coordinates. If the Cartesian coordinate system is used, they are X, Y coordinates.
- **NValues** and **Value**: **Value** is an array of values that are assigned to the polygon and **NValues** is the size of the Value array. The Value array may contain values of -999999999, which indicates a NotDefinedValue. As given below, the size and contents of the Value array vary with the parameter/package that is being edited. Refer to Section 3.9.1.1 for details about package-specific data.
 - For Layer Top Elevation, Layer Bottom Elevation, IBOUND, ICBUND, Starting Hydraulic Head, Effective Porosity, Retardation Factor, Bulk Density, Sand Box 1, Sand Box 2, Sand Box 3, Wetting Threshold, and all data of the ZoneBudget and MT3D/SEAWAT menus: NValues = 1 and Value[1] contains the respective value of these properties.
 - For aquifer properties, such as Horizontal Hydraulic Conductivity, Vertical Hydraulic Conductivity, Specific Storage, Horizontal Anisotropy, Vertical Anisotropy, Transmissivity, Vertical Leakance, Storage Coefficient, and Specific Yield: NValues = 2.
 - Value[1] = the respective aquifer property value.
 - Value[2] = Parameter Number [-].
 - **CHD**: NValues = 5.
 - Value[1] = Flag [-].
 - Value[2] = Start Head (h_s) [L].
 - Value[3] = End Head (h_e) [L].
 - Value[4] = Head Conversion [-] (SEAWAT) [-].
 - Value[5] = Water Density (d) [ML-3] (SEAWAT).
 - **DRN**: NValues = 4.

- Value[1] = Drain conductance (C_d) per unit area [T^{-1}].
- Value[2] = Drain elevation (h_d) [L].
- Value[3] = Parameter Number [-].
- Value[4] = Elevation of drain bottom [L].
- **DRT1:** NValues = 8.
 - Value[1] = Drain conductance (C_d) per unit area [T^{-1}].
 - Value[2] = Drain elevation (h_d) [L].
 - Value[3] = Column number of the return flow recipient cell [-].
 - Value[4] = Row number of the return flow recipient cell [-].
 - Value[5] = Layer number of the return flow recipient cell [-].
 - Value[6] = Return flow proportion [-].
 - Value[7] = Parameter Number [-].
 - Value[8] = Elevation of drain bottom [L].
 - Value[9] = Return flow On/Off [-]. 1= on; 0 =off.
- **ETS1:** NValues = $5 + 2 \times (NS - 1)$, where NS is the number of segments.
 - Value[1] = Maximum ET Flux (ETSR) [LT^{-1}].
 - Value[2] = Elevation of ET Surface (ETSS) [L].
 - Value[3] = ET Extinction Depth (ETSX) [L].
 - Value[4] = Layer Indicator (IETS) [-].
 - Value[5] = Parameter Number [-].
 - Value[5 + i] = PXDP [-] of the i-th segment.
 - Value[5 + i * 2] = PETM [-] of the i-th segment.
- **EVT:** NValues = 5
 - Value[1] = Maximum ET Flux (EVTR) [LT^{-1}].
 - Value[2] = Elevation of ET Surface (SURF) [L].
 - Value[3] = ET Extinction Depth (ETDP) [L].
 - Value[4] = Layer Indicator (IEVT) [-].
 - Value[5] = Parameter Number [-].
- **FHB1:** NValues = 2
 - Value[1] = boundary type
 - 0: none
 - 1: specified head
 - 2: specified flow
 - Value[2] = Cell Face (IAUX)
 - 0: Internal distributed source/sink
 - 1: Left
 - 2: Right
 - 3: Front
 - 4: Rear
 - 5: Bottom
 - 6: Top

- 7: Lateral Faces
- **GHB:** NValues = 5
 - Value [1] = Boundary conductance per unit area [T^{-1}].
 - Value [2] = Hydraulic Head on External Source (HB) [L].
 - Value[3] = GHB Elevation (SEAWAT) [L].
 - Value[4] = Water Density at GHB (SEAWAT) [L].
 - Value[5] = Parameter Number [-].
- **IBS:** NValues = 5
 - Value[1] = Preconsolidation Head (HC) [L].
 - Value[2] = Elastic Storage Factor (Sfe) [-].
 - Value[3] = Inelastic Storage Factor (Sfv) [-].
 - Value[4] = Starting Compaction (COM) [L].
 - Value[5] = Parameter Number [-].
- **RCH:** NValues = 3
 - Value[1] = Recharge Flux (RECH) [LT^{-1}].
 - Value[2] = Layer Indicator (IRCH) [-].
 - Value[3] = Parameter Number [-].
- **RES1:** NValues = 6
 - Value[1] = Reservoir Number (IRES) [-].
 - Value[2] = Land Surface Elevation (Bres) [L].
 - Value[3] = Vertical Hydraulic Conductivity (HCres) [LT^{-1}].
 - Value[4] = Reservoir Bed Thickness (Rbthck) [L].
 - Value[5] = Layer Number (IRESL) [-].
 - Value[6] = Parameter Number [-].
- **RIV:** NValues = 6
 - Value[1] = Riverbed conductance per unit area C_{riva} [T^{-1}].
 - Value[2] = River Stage h_{riv} [L].
 - Value[3] = Elevation of the Riverbed Bottom (B_{riv}) [L].
 - Value[4] = Thickness of Riverbed Sediments (SEAWAT) [L].
 - Value[5] = River Water Density (SEAWAT) [ML^{-3}].
 - Value[6] = Parameter Number [-].
- **SWI2:** NValues = 10.
 - Value[1] = ISOURCE.
 - Value[2] = Initial elevation of Zeta surface 1 [L].
 - Value[3] = Initial elevation of Zeta surface 2 [L].
 - Value[4] = Initial elevation of Zeta surface 3 [L].
 - Value[5] = Initial elevation of Zeta surface 4 [L].
 - Value[6] = Initial elevation of Zeta surface 5 [L].
 - Value[7] = Initial elevation of Zeta surface 6 [L].
 - Value[8] = Initial elevation of Zeta surface 7 [L].
 - Value[9] = Initial elevation of Zeta surface 8 [L].

- Value[10] = Initial elevation of Zeta surface 9 [L].
- **UZF1:** NValues = 11.
 - Value[1] = Aerial Extent (IUZFBND).
 - Value[2] = Vertical Saturated Hydraulic Conductivity (VKS) [LT^{-1}].
 - Value[3] = Brooks-Corey Epsilon (EPS) [-].
 - Value[4] = Saturated Water Content (THTS) [L^3L^{-3}].
 - Value[5] = Residual Water Content (THTR) [L^3L^{-3}].
 - Value[6] = Initial Water Content (THTI) [L^3L^{-3}].
 - Value[7] = Infiltration Rate (FINF) [LT^{-1}].
 - Value[8] = Potential ET (PET) [LT^{-1}].
 - Value[9] = ET Extinction Depth (EXTDP) [L].
 - Value[10] = ET Extinction Water Content (EXTWC) [L^3L^{-3}].
 - Value[11] = Parameter Number [-].
- **WEL:** NValues = 3.
 - Value[1] = Flow Rate per unit area [LT^{-1}].
 - Value[2] = Fluid Density (SEAWAT) [ML^{-3}].
 - Value[3] = Parameter Number [-].
- **NVars** and **Var** are used to define the time series of prescribed flux or head in the FHB1 package. **Var** is the time series array and **NVars** is the size of array.
- **NConcs** and **Conc** are used to define the concentration values associated with species. **Conc** an array containing concentration data of individual species and **NConcs** is the size of the Conc array. NConcs is limited to a maximum of 100.

5.1.12. Polyline File

A Polyline file is a text file with the following format. You can export or import polyline files via the Data Editor, see Section 3.4.2.2 for details.

File Format

1. #PM_POLYLINES
2. Version
3. NPolylines
4. (repeat the following data block, i.e., data 4 – 15, NPolylines times; a block for a polyline)
5. DataVersion
6. Name
7. IsOn, AutoLayerNumber, ShowVertices, Priority
8. LineWidth, Red, Green, Blue
9. NProperties
10. Property (an array of NProperties values separated by space)
11. NP
12. (repeat the following data block NP times; a block for a vertex)

13. VertexDataVersion
14. Vertex_id
15. Att1
16. Longitude/X, Latitude/Y
17. NValues
18. Value (an array of NValues values separated by space or in different lines)
19. NVars
20. Var (an array of NVars values separated by space or in different lines)
21. NConcs
22. Conc (an array of NConcs values separated by space or in different lines)
23. Naux1
24. Aux1 (an array of Naux1 values separated by space or in different lines)
25. Naux2
26. Aux2 (an array of Naux2 values separated by space or in different lines)

Description

- **#PM_POLYLINES:** The first line holds the text string “#PM_POLYLINES” that serves as the file format identifier.
- **Version** is the version number that is reserved for future use. The value for the current version is 1.04
- **NPolylines** is the number of polylines.
- **DataVersion** is the version number of the data block. The value for the current DataVersion is 1.
- **Name** is the name of the polyline.
- **IsOn** is a value equals to either 1 or 0. The value of 1 means that the polyline is turned on; 0 means off.
- **AutoLayerNumber** is a value equals to either 1 or 0. The value of 1 means that when applying the polyline to cells, the auto layer number feature is on; 0 means off.
- **ShowVertices** is a value equals to either 1 or 0. The value of 1 means that the vertices of the polyline is displayed; 0 means not displayed.
- **Priority:** When applying overlapping polyline to cells, the polyline with the smallest priority value has the highest priority.
- **LineWidth:** This value is reserved for future use. It should be 2.
- **Red, Blue, Green:** These values are reserved for future use. They should range between 0 and 255.
- **NProperties and Properties.** **NProperties** is number of polyline properties. **Properties** is the array of polyline properties. Polyline properties are used by the following packages. For all other packages or aquifer properties, NProperties should be set to zero.
 - **CHD:** NProperties = 1
 - Property[1] defines Head Conversion Option for SEAWAT as follows

- 0: Use calculated water density at the cell
 - 1: Use the specified water density value
 - 2: Use the average of the calculated water density of the overlying cells
 - 3: Input head values are reference head. No conversion
 - **DRT:** NProperties = 1
 - Property[1] controls whether Return flow is On or Off.
 - 0: Return flow is off.
 - 1: Return flow is on.
 - **FHB:** NProperties = 2
 - Property[1] is the boundary type
 - 0: none
 - 1: specified head
 - 2: specified flow
 - Property[2] defines the Cell Face (IAUX)
 - 0: Internal distributed source/sink
 - 1: Left
 - 2: Right
 - 3: Front
 - 4: Rear
 - 5: Bottom
 - 6: Top
 - 7: Lateral Faces
 - **SFR2:** NProperties = 1
 - Property[1] is the Segment Number
 - **STR:** NProperties = 2
 - Property[1] is the Segment Number
 - Property[2] is the Segment Inflow.
- **NP:** Number of vertices of the polyline. A polyline must have at least 3 vertices. The number of vertices must be odd. The odd numbered vertices are big white vertices (see Section 3.4.2.3 for the vertex types), the even numbered vertices are small gray vertices.
- **VertexDataVersion** is the version number of the vertex block. The value for the current VertexDataVersion is 1.01.
- **Vertex_id**, introduced since VertexDataVersion 1.01, is the unique identifier of the vertex.
- **Att1**, introduced since VertexDataVersion 1.04, is an attribute of the vertex.
- **Longitude/X, Latitude/Y** are coordinates of the vertex. If the Geographic coordinate system is used, they are expressed in the WGS 84 geographic coordinates. If the Cartesian coordinate system is used, they are X, Y coordinates.
- **NValues** and **Value:** **Value** is an array of values that are assigned to the vertex. NValues is the size of the Value array. Note that a Value array may only be assigned to odd numbered vertices. Values that are assigned to even numbered vertices will be discarded. The Value

array may contain values of -999999999, which indicates a NotDefinedValue. As given below, the size and contents of the Value array vary with the parameter/package that is being edited. Refer to Section 3.9.1.1 for details about package-specific data.

- For Layer Top Elevation, Layer Bottom Elevation, IBOUND, ICBUND, Starting Hydraulic Head, Effective Porosity, Retardation Factor, Bulk Density, Sand Box 1, Sand Box 2, Sand Box 3, Wetting Threshold, and all data of the ZoneBudget and MT3D/SEAWAT menus: NValues = 1 and Value[1] contains the respective value of these properties.
- For aquifer properties, such as Horizontal Hydraulic Conductivity, Vertical Hydraulic Conductivity, Specific Storage, Horizontal Anisotropy, Vertical Anisotropy, Transmissivity, Vertical Leakage, Storage Coefficient, and Specific Yield: NValues = 2.
 - Value[1] = the respective aquifer property value.
 - Value[2] = Parameter Number [-].
- **DRN:** NValues = 4.
 - Value[1] = Drain conductance per unit length [LT^{-1}].
 - Value[2] = Drain elevation (h_d) [L].
 - Value[3] = Parameter Number [-].
 - Value[4] = Elevation of drain bottom [L].
- **DRT1:** NValues = 8.
 - Value[1] = Drain conductance per unit length [LT^{-1}].
 - Value[2] = Drain elevation (h_d) [L].
 - Value[3] = Column number of the return flow recipient cell [-].
 - Value[4] = Row number of the return flow recipient cell [-].
 - Value[5] = Layer number of the return flow recipient cell [-].
 - Value[6] = Return flow proportion [-].
 - Value[7] = Parameter Number [-].
 - Value[8] = Elevation of drain bottom [L].
- **ETS1:** NValues = $5 + 2 \times (NS - 1)$, where NS is the number of segments.
 - Value[1] = Maximum ET Flux (ETSR) [LT^{-1}].
 - Value[2] = Elevation of ET Surface (ETSS) [L].
 - Value[3] = ET Extinction Depth (ETSX) [L].
 - Value[4] = Layer Indicator (IETS) [-].
 - Value[5] = Parameter Number [-].
 - Value[5 + i] = PXDP [-] of the i-th segment.
 - Value[5 + i * 2] = PETM [-] of the i-th segment.
- **EVT:** NValues = 5
 - Value[1] = Maximum ET Flux (EVTR) [LT^{-1}].
 - Value[2] = Elevation of ET Surface (SURF) [L].
 - Value[3] = ET Extinction Depth (ETDP) [L].
 - Value[4] = Layer Indicator (IEVT) [-].

- Value[5] = Parameter Number [-].
- **FHB1:** NValues = 0
- **GHB:** NValues = 5
 - Value[1] = Boundary conductance per unit length [LT^{-1}].
 - Value[2] = Hydraulic Head on External Source (HB) [L].
 - Value[3] = GHB Elevation (SEAWAT) [L].
 - Value[4] = Water Density at GHB (SEAWAT) [L].
 - Value[5] = Parameter Number [-].
- **HFB:** NValues = 2
 - Value[1] = Hydraulic characteristic [T^{-1}]
 - Value[2] = Parameter Number [-].
- **IBS:** NValues = 5
 - Value[1] = Preconsolidation Head (HC) [L].
 - Value[2] = Elastic Storage Factor (Sfe) [-].
 - Value[3] = Inelastic Storage Factor (Sfv) [-].
 - Value[4] = Starting Compaction (COM) [L].
 - Value[5] = Parameter Number [-].
- **RCH:** NValues = 3
 - Value[1] = Recharge Flux (RECH) [LT^{-1}].
 - Value[2] = Layer Indicator (IRCH) [-].
 - Value[3] = Parameter Number [-].
- **RIV:** NValues = 6
 - Value[1] = Riverbed conductance per unit length C_{rivl} [LT^{-1}].
 - Value[2] = River Stage h_{riv} [L].
 - Value[3] = Elevation of the Riverbed Bottom (B_{riv}) [L].
 - Value[4] = Thickness of Riverbed Sediments (SEAWAT) [L].
 - Value[5] = River Water Density (SEAWAT) [ML^{-3}].
 - Value[6] = Parameter Number [-].
- **SFR2:** NValues = 16.
 - Value[1] = Stream slope (SLOPE) [-].
 - Value[2] = Top Elevation of the Streambed (STRTOP) [L]:
 - Value[3] = Thickness of the Streambed (STRTHICK) [L].
 - Value[4] = Streambed Hydraulic Conductivity (STRHC1) [LT^{-1}].
 - Value[5] = Saturated Volumetric Water Content (THTS) [-].
 - Value[6] = Initial Volumetric Water Content (THTI) [-]:
 - Value[7] = Brooks-Corey Exponent (EPS) [-].
 - Value[8] = Vertical Saturated Hydraulic Conductivity (UHC) [LT^{-1}].
 - Value[9] = Parameter Number (PEST) [-].
 - Value[10] – Value[16]: reserved and not used.
- **STR:** NValues = 8. Assign -999999999 to Value[5], Value[6], and Value[7], if *Stream Stage in Reaches* is not to be calculated.

- Value[1] = Stream Stage (h_{str}) [L].
- Value[2] = Streambed Conductance (C_{str}) [L^2T^{-1}].
- Value[3] = Elevation of Streambed Top (T_{str}) [L].
- Value[4] = Elevation of Streambed Bottom (B_{str}) [L].
- Value[5] = Width of the Stream Channel (W_{str}) [L].
- Value[6] = Slope of the Stream Channel (S_{str}) [-].
- Value[7] = Manning's Roughness Coefficient (n) [$TL^{-1/3}$].
- Value[8] = Parameter Number [-].
- **SWI2:** NValues = 10.
 - Value[1] = ISOURCE.
 - Value[2] = Initial elevation of Zeta surface 1 [L].
 - Value[3] = Initial elevation of Zeta surface 2 [L].
 - Value[4] = Initial elevation of Zeta surface 3 [L].
 - Value[5] = Initial elevation of Zeta surface 4 [L].
 - Value[6] = Initial elevation of Zeta surface 5 [L].
 - Value[7] = Initial elevation of Zeta surface 6 [L].
 - Value[8] = Initial elevation of Zeta surface 7 [L].
 - Value[9] = Initial elevation of Zeta surface 8 [L].
 - Value[10] = Initial elevation of Zeta surface 9 [L].
- **UZF1:** NValues = 11.
 - Value[1] = Aerial Extent (IUZFBND).
 - Value[2] = Vertical Saturated Hydraulic Conductivity (VKS) [LT^{-1}].
 - Value[3] = Brooks-Corey Epsilon (EPS) [-].
 - Value[4] = Saturated Water Content (THTS) [L^3L^{-3}].
 - Value[5] = Residual Water Content (THTR) [L^3L^{-3}].
 - Value[6] = Initial Water Content (THTI) [L^3L^{-3}].
 - Value[7] = Infiltration Rate (FINF) [LT^{-1}].
 - Value[8] = Potential ET (PET) [LT^{-1}].
 - Value[9] = ET Extinction Depth (EXTDP) [L].
 - Value[10] = ET Extinction Water Content (EXTWC) [L^3L^{-3}].
 - Value[11] = Parameter Number [-].
- **WEL:** NValues = 3.
 - Value[1] = Flow Rate per unit length [L^2T^{-1}].
 - Value[2] = Fluid Density (SEAWAT) [ML^{-3}].
 - Value[3] = Parameter Number [-].
- **NVars** and **Var** are used to define the time series of prescribed flux or head in the FHB1 package. **Var** is the time series array and **NVars** is the size of array.
- **NConcs** and **Conc** are used to define the concentration values associated with species. **Conc** an array containing concentration data of individual species and **NConcs** is the size of the Conc array. NConcs is limited to a maximum of 100.
- **Naux1** and **Aux1** are introduced in VertexDataVersion 1.04 and are currently only used by SFR2 package to define the initial concentration values associated with species. Those

values are intended to be used by the SFT Package. **Aux1** an array containing concentration data of individual species and **Naux1** is the size of the **Aux1** array. **Naux1** is limited to a maximum of 100.

- **Naux2** and **Aux2** are introduced in VertexDataVersion 1.04 and are currently only used by SFR2 package to define the stream dispersion values associated with species. Those values are intended to be used by the SFT Package. **Aux2** an array containing concentration data of individual species and **Naux2** is the size of the **Aux2** array. **Naux2** is limited to a maximum of 100.

5.1.13. Reservoir Stage Time Series

A Reservoir Stage Time Series file is a text file that contains the stage measurements of reservoirs that are used by the Reservoir Package (RES1). You can export or import reservoir stage time series files via the Reservoir Stage Time Series dialog box (Figure 86).

File Format

1. #RESERVOIR_STAGE_SERIES
2. Version
3. NRES
4. (repeat the following block NRES times)
5. ReservoirNumber, ReservoirName
6. N
7. (repeat the following line N times)
8. Time, Stage

Description

- **#RESERVOIR_STAGE_SERIES:** The first line holds the text string “#RESERVOIR_STAGE_SERIES” that serves as the file format identifier.
- **Version** is the version number that is reserved for future use. The value for the current version is 1.
- **NRES** is the number of reservoirs.
- **ReservoirNumber** and **ReservoirName** are the number and name of the reservoir, respectively.
- **N** is the number of data points that will follow.
- **Time** and **Stage** are the simulation time and the reservoir stage, respectively.

5.1.14. Series File

A Series file is a text file that contains a series of numbers. Many dialog boxes of PM can load or save Series files.

File Format

1. #SERIES
2. Version

3. N
4. v(1) v(2) v3 v(N)

Description

- **#SERIES:** The first line holds the text string "#SERIES" that serves as the file format identifier.
- **Version** is the version number that is reserved for future use. The value for the current version is 1.
- **N** is the number of values in each of the following lines. Most dialog boxes of PM use N = 2 for time series.
- **v(1), v(2), ..., v(N)** are the values of the series. These values must be separated by a space or comma, or in different lines. Repeat this line as often as needed.

Example

The following is an example showing a (time) series file. N is set to 2 indicating that the first two values contained in the lines will be used. N is followed by five lines; each contains a time and a value.

```
#SERIES
1
2
0 45.2
30 46
60 46.2
90 46.1
120 46.3
```

5.1.15. SURFER ASCII GRD File

A SURFER GRD file may only be imported to regularly spaced model grids. PM only accepts SURFER ASCII GRD files. Binary GRD files cannot be imported.

File Format

```
DSAA
ncols nrows
Xmin Xmax
Ymin Ymax
Zmin Zmax
Z11 Z12 Z13 ... Z1n
Z21 Z22 Z23 ... Z2n
Z31 Z32 Z33 ... Z3n
...
ZM1 ZM2 ZM3 ... ZMn
```

Description

1. DSAA: The first line holds the ASCII (text) string "DSAA" that serves as the file format identifier.
1. ncols and nrows are the numbers of columns and rows, respectively.
2. Xmin Xmax are the minimum and maximum x values, respectively.
3. Ymin Ymax are the minimum and maximum y values, respectively.
4. Zmin Zmax are the minimum and maximum z values, respectively.
5. The remainder of the file lists the matrix values, starting at the lower-left corner (i.e., Z11). These numbers are delimited using a single space character. Note that the first row is the one associated with the minimum value of y and represents the bottom edge of your map. The last row is the top edge of the map. The leftmost value in each row is that associated with the minimum x, the rightmost goes with the maximum x.

Example

The following is an example showing an ASCII Raster file for a grid with 4 columns and 3 rows.

```
DSAA
4 3
-117.73 -117.71
33.451 33.471
11 42
23 41 31 19
42 35 23 21
11 12 34 23
```

5.1.16. Tabular Inflow File

A tabular inflow file is a comma-separated values text file stored in a table structured format and contains the dates and inflow values of stream segments.

File Format

The first line of a tabular inflow file contains the column headers. The first three items of the first line are year, month, and day, respectively. The following items are segment numbers.

The remainder of the file contain dates and inflow data. The first three items of a line are year, month, and day that define the date of the inflow data. The following items of the line contain the inflow data pertaining to the respective segments (specified in the column headers) for the given date. The inflow data must be specified in the unit of $[L^3/T]$ that is consistent with the length and time units of the model.

Example

The following is an example showing a Tabular Inflow file containing inflow data to three stream segments (with the segment number 4, 6, and 9) of four dates.

Year,	Month,	Day,	4,	6,	9
2020,	12,	1,	810,	198,	150
2021,	1,	1,	780,	190,	160
2021	2,	1,	785,	196,	164
2021	3,	1,	790,	201,	161

5.1.17. Temporal Discretization File

A temporal discretization file is a text file that contains the temporal discretization data that can be exported from or imported to the Time dialog box (see Section 3.7.1).

File Format

1. #TemporalDiscretization
2. Version
3. NPER TimeUnit
4. (the following line repeats NPER times)
5. IsSteady, PERLEN, NSTP, TSMULT, DTO, MXSTRN, TTSMULT, TTSMAX, FLAG

Description

- **#TemporalDiscretization:** The first line holds the text string “#TemporalDiscretization” that serves as the file format identifier.
- **Version** is the version number that is reserved for future use. The value for the current version is 1.
- **NPER** is the number of stress periods
- **TimeUnit** is an integer between 0 and 4: 0 = Second; 1 = Minute; 2 = Hour; 3 = Day; 4 = Year.
- **IsSteady** > 0 if the stress period is steady state. **IsSteady** = 0 for a transient stress period.
- **PERLEN** is the length of the stress period, in the unit of TimeUnit.
- **NSTP** is the number of time steps within a stress period.
- **TSMULT** is the time step multiplier, which is the ratio of the length of each time step to that of the preceding time step.
- **DTO** is the initial transport step length within each time step of the flow solution.
- **MXSTRN** is the maximum number of transport steps allowed within one time step of the flow solution.
- **TTSMULT** is the multiplier for successive transport steps within a flow time step.
- **TTSMAX** is the maximum transport step size.
- **FLAG** is an integer reserved for future use. It should be set as zero.

5.1.18. Well File

A Well File⁹⁰ contains the meta data of wells (name, location, and screen intervals), the pumping time series, and optionally the concentration values associated with injection wells. Well files may be imported to the Well (WEL) Package or the Multi-Node Well 2 (MNW2) Package with *Tools > Package > Append Wells* (see Section 3.6.21.) The imported data will be appended to the existing data of the MNW2 or WEL Package.

When importing a Well File to the MNW2 Package, it is assumed that the well nodes are defined by screen intervals, and the loss type is THIEM with the default well radius $R_w = 0.2$. The value of R_w may optionally be defined for individual wells.

When importing a Well File to the WEL Package, pumping/Injection rate of a well is prorated to the vertical column of cells where the well is located. The proration is calculated based on the following equation.

$$Q_i = \frac{Q_w \times HK_i \times L_i \times F_i}{\sum_{i=1}^{NLAY} (HK_i \times L_i \times F_i)} \quad (58)$$

where

i is the layer number,

Q_w is the pumping/injection rate of the well,

Q_i is the pumping/injection rate that is assigned to the cell in the i^{th} layer,

$NLAY$ is the number of model layers,

HK_i is the hydraulic conductivity of the cell in the i^{th} layer,

L_i is the total length of saturated well screens or well screens within the i^{th} layer. See [Head] in Description below for details.

$F_i = 1$ if the cell in the i^{th} layer is a variable head (active) cell; and $F_i = 0$ if no-flow (inactive).

File Format

1. #WellFile [Depth] [Head] [IncludeNoPumping] [Radius]
2. WellName, Latitude/Y, Longitude/X, [Rw], [Top1, Bottom1,] [Top2, Bottom2] [##]...
(repeat this line for each well)
3. #Pumping
4. p, Q1_p, Q2_p, ..., Qn_p
(repeat this line for each period at which pumping/injection are to be assigned)
5. The following data are optional and are needed only if you need to assign concentration values to inject wells.
6. [#Concentration, SpeciesNumber]

⁹⁰ The Well File format is not the same as the format of the Well Package input file. The latter is defined in the user guide of MODFLOW-2005 (Harbaugh, 2005).

7. [p, C1_p, C2_p, ..., Cn_p]

(repeat this line for each period at which the concentration values of the corresponding injection are to be assigned)

Description

- **#WellFile:** The first line of the Well File holds the text string “#WellFile” that serves as the file format identifier.
- **[Depth]:** This is an optional case-insensitive keyword. If this keyword is present, the screen intervals are defined by depths from the top of the first layer; otherwise, the screen intervals are defined by elevation values.
- **[Head]:** This optional case-insensitive keyword is used only when importing the file to the WEL Package. If this keyword is present, the starting hydraulic head is used to determine the saturated length of well screens, and L_i in the equation above is the total length of saturated well screens within the cell in the i^{th} layer; otherwise L_i is the total length of well screens within the cell in the i^{th} layer.
- **[IncludeNoPumping]:** This optional case-insensitive keyword is used only when importing the file to the WEL Package. If this keyword is present, wells with a pumping rate equal to zero will be imported as inactive wells; otherwise those wells will not be imported.
- **[Radius]:** This optional case-insensitive keyword is used only when importing the file to the MNW2 Package. If this keyword is present, the well radius R_w must be defined for each well. If this keyword is not present, the default value of $R_w = 2$ is applied to all wells. R_w is used by the MNW2 Package only when the Loss Type is set to THIEM, SKIN, or GENERAL.
- **Wellname** is well name. Wellname may contain spaces but may not include comma.
- **Longitude/X, Latitude/Y** are coordinates of the well. If the Geographic coordinate system is used, they are expressed in the WGS 84 geographic coordinates. If the Cartesian coordinate system is used, they are X, Y coordinates.
- **Rw:** This is the well radius used by the MNW2 Package for the loss type THIEM or SKIN. R_w is ignored by the WEL Package. If the keyword “Radius” appears in the first line of the Well File, the R_w value must be included. If the keyword *Radius* does not appear in the first line of the Well File, the R_w value may not be included.
- **Top1** and **Bottom1** defines the first screen interval from top down; **Top2** and **Bottom2** defines the second screen interval. Repeat the top/bottom pairs from top down for additional screen intervals as needed. If screen interval data are not specified, the well is assumed to be perforated in the highest active model layer.
 - If the first line of the Well File contains the case-insensitive keyword “Depth”, top and bottom values are the respective *depths* from the top of the first layer to the top and bottom of a screen interval.
 - If the first line of the Well File does not contain the case-insensitive keyword “Depth”, top and bottom values are the respective *elevations* of the top and bottom of a screen interval.

- **##** is a marker for comments. The “##” marker and the contents to the right of the marker are ignored.
- **#Pumping**. The line starting with “#Pumping” (or “#Period”) is used to terminate the well information and to denote the start of the pumping data.
- **p** is the period number at which the pumping/injection rates or concentration values are to be assigned.
- **Q1_p, Q2_p, ..., Qn_p** are the pumping/injection rate of well 1, well 2, ..., well n in the period **p**. These values must be given in the same order as the wells appear in Data 2. Use negative values for pumping; positive values for injection.
- **#Concentration, SpeciesNumber**. The line starting with “#Concentration” is used to terminate the pumping data or concentration data of another species, and to denote the start of the concentration data of the species SpeciesNumber. If SpeciesNumber is not specified, it is set to 1. This line is optional.
- **C1_p, C2_p, ..., Cn_p** are the concentration values of the speciesNumber of well 1, well 2, ..., well n in the period **p**. This line is optional. The values must be given in the same order as the wells appear in Data 2. Concentration values must be greater or equal to zero and are only effective at injection wells. Negative concentration values and concentration values that are assigned to pumping wells are ignored.

Example

Table 4 shows the coordinates, screen interval elevations, and pumping/injection rates of three wells. For Well 3, the injection concentration of species 1 are 100 and 200 for period 1 and 2, respectively; and the injection concentration of species 2 are 12 and 6 for period 1 and 2, respectively.

The corresponding Well File for these wells is as follows.

```
#WellFile
Well 1, 34.12, -117.33, 100, 90, 60, 40
Well 2, 34.56, -117.67, 80, 50
Well 3, 34.22, -117.28, 82, 43
#Pumping
1, -50, -45, 25
2, -60, -55, 40
#Concentration, 1
1, 0, 0, 100
2, 0, 0, 200
#Concentration, 2
1, 0, 0, 12
2, 0, 0, 6
```

Table 4. Well Data Example

Well Name	Latitude	Longitude	Top1	Bottm1	Top2	Bottom2	Pumping (period 1)	Pumping (period 2)
Well 1	34.12	-117.33	100	90	60	40	-50	-60
Well 2	34.56	-117.67	80	50			-45	-55
Well 3	34.22	-117.28	82	43			25	40

5.2. Utilities

Several utility codes are used to enable PEST for model calibration. These codes process the model output files and save the processed simulated results in the formats that can be used for model calibration purposes.

5.2.1. MOD2OBS

MOD2OBS (Watermark Numerical Computing, 2020) interpolates model-generated head or drawdown data to the times and locations specified by the user and writes results to an output file. MOD2OBS requires several input files. When MOD2OBS runs, it prompts for the filenames and several input parameters.

To automate the execution process, PM creates the required input files, stores the filenames and input parameters in the `pest.mod2obsheadinput` or `pest.mod2obsdrawdowninput` files, and then writes the following commands to the `modelrun.bat` file, which is executed during a PEST iteration.

```
mod2obs.exe < pest.mod2obsheadinput
```

```
mod2obs.exe < pest.mod2obsdrawdowninput
```

File types and filenames listed in the `pest.mod2obsheadinput` or `pest.mod2obsdrawdowninput` files are given below. Refer to Groundwater Data Utilities: Part A Overview (Watermark Numerical Computing, 2020) for details of the file formats.

- Grid Specification File: `pest.gridspecification`
- Bore Coordinates File: `pest.borecoordinate`
- Bore Sample File: `pest.boresamplehead`
- Modflow Output File: `modflow.head` or `modflow.drawdown`
- MOD2OBS Output File: `pest.mod2obsheadoutput` or `pest.mod2obsdrawdownoutput`

5.2.2. KRIGING

PM implements the Kriging process with the programs `ppk2fac1` and `fac2real` (Doherty, 2020) on a layer-by-layer basis. The first program `ppk2fac1` calculates *kriging factors* from a set of pilot points to the model cells in a specific layer. The second program `fac2real` uses the `ppk2fac1`-generated *kriging factors* to interpolate the parameter values of the pilot points to the model

cells in that layer. The Kriging interpolation is based on user-defined variograms (see Section 3.4.2.4.5).

In addition to `ppk2fac1` and `fac2real`, the program `ppkreg1` (Doherty, 2020) may be used to add regularization information to a PEST control file, and the program `ppcov` (Doherty, 2020) may be used to generate covariance matrix files for a PESTPP-IES run. Refer to Section 3.9.5.4 for details.

5.2.3. FHBADJUST

FHBADJUST enables PEST to adjust user specified flow and head values in the FHB1 Package. Prior to a PEST run, PM creates the `modflow.fhb0` file (containing the user specified flow and head values) and the `pest.fhbm` template file (containing place holders for specified flow multipliers and/or specified head addends). During a PEST run, the `modflow.fhbm` file is created by populating the PEST-estimated multipliers and/or addends to the respective place holders in the template file. FHBADJUST then generates the `modflow.fhb` file based on the `modflow.fhb0` and `modflow.fhbm` files. The `modflow.fhb` file is used by the FHB1 Package of MODFLOW.

To automate the process, PM writes the following command to the `modelrun.bat` file, which is executed during a PEST iteration.

```
fhbadjust.exe modflow.fhb0 modflow.fhbm modflow.fhb
```

5.2.4. UPDATEWEIGHTS

UPDATEWEIGHTS updates the layer weight values of observation points during a PEST run. The updated layer weight value w' for an observation point in a specific layer is calculated as $w' = (w / HK) * HK'$, where w is the initial layer weight value, HK is the initial horizontal hydraulic conductivity prior to a PEST run, and HK' is the updated horizontal hydraulic conductivity in a PEST iteration. The updated layer weight values for all layers are normalized so that their sum equals 1.

PM writes the following commands to the `modelrun.bat` file, which is executed during a PEST iteration. These commands update layer weight values of head and drawdown observations, respectively. The value of N in these commands are replaced by the number of model layers.

```
updateweights.exe pest.headborelayer pest.headboreinfo N
```

```
updateweights.exe pest.ddownborelayer pest.ddownboreinfo N
```

PM creates the following files used by those commands prior to a PEST run:

- Bore Information File for head observations: `pest.headboreinfo`. This file contains three columns, namely `PointName~Layer`, W , and D .
 - `PointName` is the name of an observation point.
 - `Layer` is the model layer where the layer weight value of the observation point is greater than 0. The corresponding model cell is a “perforated cell”.

- W is the layer weight value of the respective model layer.
- D is the default head value and it is equal to the bottom of the lowest perforated cell of an observation point. If all perforated cells of an observation point drop dry during a PEST iteration, LAYERWEIGHT sets this value as the layer-weighted head.
- Bore Layer Weight File for head observations: pest.headborelayer. This file contains the data of observation points that have head observations. The data are listed in six columns, namely PointName, Layer, TKFlag, CellIndex, Ratio, and Thickness.
 - PointName is the name of an observation point.
 - Layer is the model layer where the layer weight value of the observation point is greater than 0.
 - TKFlag is 1 if T of the corresponding model layer in the Layer Properties dialog box (Figure 50) is User-Specified.
 - CellIndex is the index of the cell where the observation point lies. The index of a cell located in [row, column] is calculated as $\text{CellIndex} = [\text{column} + \text{row} * \text{number of columns}]$.
 - Ratio is calculated as $\text{Ratio} = [\text{initial layer weight} / \text{HK}]$.
 - Thickness is the thickness of the cell pointed to by CellIndex.

Similarly, the following two files are created for drawdown observations:

- Bore Information File for drawdown observations: pest.ddownboreinfo: This file contains three columns, namely PointName~Layer, W, and D.
 - PointName is the name of the observation point with multi-layer observations.
 - Layer is the model layer where the layer weight value of the observation point is greater than 0. The corresponding model cell is a “perforated cell”.
 - W is the layer weight value of the respective model layer.
 - D is the default drawdown value and it is equal to maximum difference between the starting heads of perforated cells and the bottom of the lowest perforated cell of an observation point. If all perforated cells of an observation point drop dry during a PEST iteration, LAYERWEIGHT sets this value as the layer-weighted drawdown.
- Bore Layer Weight File for drawdown observations: pest.ddownborelayer. Similar to the pest.headborelayer given above, this file contains the data of observation points that have drawdown observations.

5.2.5. LAYERWEIGHT

LAYERWEIGHT reads the output file of MOD2OBS (containing head or drawdown values in specific locations, layers, and times) and a Bore Information file (containing observation point, layer, and layer weight), then calculates layer-weighted head or drawdown values.

PM creates the required input files and then writes the following commands to the modelrun.bat file, which is executed during a PEST iteration. These commands calculate layer-weighted head or drawdown values, respectively.

layerweight.exe pest.headboreinfo pest.mod2obsheadoutput pest.headoutput

layerweight.exe pest.ddownboreinfo pest.mod2obsdrawdownoutput pest.drawdownoutput

Input file types and respective filenames to LAYERWEIGHT are listed below.

- Bore Information File: pest.headboreinfo or pest.ddownnboreinfo. See Section 5.2.4 above for details.
- MOD2OBS Output File: pest.mod2obsheadoutput or pest.mod2obsdrawdownoutput. See Section 5.2.1 above for details.
- LAYERWEIGHT Output File: The calculated layer-weighted head or drawdown values are stored in the pest.headoutput or pest.drawdownoutput, respectively.

5.2.6. LAKEGAGE

LAKEGAGE reads the output file of the Gage Package (containing stage values over time in lakes that are set as gaging stations), a Lake Observation Point file (containing observation point name, lake number, and weight), and a file containing the times of interest, then calculates the weighted stage values at the times of interest for each observation point.

PM creates the required input files and then writes the following commands to the modelrun.bat file, which is executed during a PEST iteration.

lakegage.exe modflow.lakegageoutput pest.lakeobspt pest.lakestageobstimes pest.lakeoutput

Input file types and respective filenames to LAKEGAGE are listed below.

- Gage Package Output File: modflow.lakegageoutput
- Lake Observation Point File: pest.lakeobspt
- Times of Interest File: pest.lakestageobstimes
- STREAMGAGE Output File: pest.lakeoutput

5.2.7. STREAMGAGE

STREAMGAGE reads the output file of the Gage Package (containing streamflow values over time in stream reaches that are designated as gaging stations), a Stream Reach file (containing observation point name, segment, reach, and weight), and a file containing the times of interest, then calculates the weighted streamflow values at the times of interest for each observation point.

PM creates the required input files and then writes the following commands to the modelrun.bat file, which is executed during a PEST iteration.

streamgage.exe modflow.streamgageoutput pest.streamreach pest.streamflowobstimes pest.streamoutput

Input file types and respective filenames to STREAMGAGE are listed below.

- Gage Package Output File: modflow.streamgageoutput
- Stream Reach File: pest.streamreach

- Times of Interest File: pest.streamflowobstimes
- STREAMGAGE Output File: pest.streamoutput

5.2.8. REAL2MATRIX

REAL2MATRIX expects a Zone Array file (containing pilot point zones), a Kriging result file generated by the fac2real program (Doherty, 2020), and a 2D matrix file (containing the aquifer property, such as HK, of a model layer). REAL2MATRIX copies the Kriging results of the model cells pertaining to a pilot point zone to the 2D matrix file. The values of the cells where interpolation does not take place remain unchanged.

PM creates the required input files and then writes the following commands to the modelrun.bat file, which is executed during a PEST iteration.

Call RunReal2Matrix.bat

The RunReal2Matrix.bat file runs the real2matrix program for each aquifer parameter in each model layer that includes pilot points. The batch files and all related files are in the Kriging subfolder of the model data folder, except for the 2D matrix files that are in the Matrix subfolder.

5.3. Input and Output Data Files of the Supported Models and Utilities

This section lists the names of the input and output files of the supported model codes and packages. Refer to the documents of the model codes for details about the file formats. By default, all input and output files of a model are stored in the *model data folder*. The name of this folder is ModelID and it can be found in the same folder as the PM file. In addition to the input and output files given in this section, PM will create and maintain several binary files in the “binary” subfolder of the model data folder. These binary files are used by PM to display cell information on the Properties window.

5.3.1. MODFLOW

5.3.1.1. Input Files

Basic Package	modflow.bas6
Block-Centered Flow Package	modflow.bcf6
Discretization	modflow.dis
Direct Solver Package	modflow.de4
Drain Package	modflow.drn
Drain with Return Flow Package	modflow.drt
Evapotranspiration Package	modflow.evt
Evapotranspiration Segments Package	modflow.ets
Flow and Head Boundary Package	modflow.fhb
Gage Package	modflow.gage

General-Head Boundary Package	modflow.ghb
Generic Multigrid Solver Package	modflow.gmg
Horizontal-Flow Barrier Package	modflow.hfb6
Interbed-Storage Package	modflow.ibs
Lake Package	modflow.lak
Layer-Property Flow Package	modflow.lpf
LMT Package	modflow.lmt
Multi-Node Well 1 Package	modflow.mnw1
Multi-Node Well 2 Package	modflow.mnw2
Newton Solver Package	modflow.nwt
Name File	modflow.nam
Output Control	modflow.oc
Preconditioned Conjugate Gradient (PCG) Solver Package	modflow.pcg
PCG Solver with Improved Nonlinear Control Package	modflow.pcgm
Recharge Package	modflow.rch
Reservoir Package	modflow.res1
River Package	modflow.riv
Seawater Intrusion 2 Package	modflow.swi
Stream (STR) Package	modflow.str
Streamflow-Routing (SFR2) Package	modflow.sfr2
Strongly Implicit Procedure Solver Package	modflow.sip
Subsidence and Aquifer-System Compaction Package	mudflow.sub
Time Variant Specified Head	modflow.chd
Unsaturated-Flow Package	modflow.uzf
Upstream Weighting Package	modflow.upw
Well Package	modflow.wel

5.3.1.2. Output Files

Run Listing	modflow.list
Drawdown	modflow.drawdown
Flow (cell-by-cell flow terms)	modflow.flow
Flow-transport link	modflow.linkmt3d
Gage Package	modflow.streamgageoutput
	modflow.lakegageoutput
Hydraulic head	modflow.head
Interbed-Storage Package	modflow.interbed
Multi-Node Well Information (MNWI) Package	modflow_mnwi_bynode.txt
	modflow_mnwi_qsum.txt
	modflow_mnwi_wel1.txt
Seawater Intrusion 2 Package	modflow.swi2zeta
Streamflow-Routing (SFR2) Package	modflow.sfr2.list

Subsidence and Aquifer-System Compaction Package⁹¹

modflow.subsidence,
modflow.compaction,
modflow.zdisplacement

5.3.2. MT3DMS/MT3D-USGS

5.3.2.1. Input Files

Advection Package

mt3d.adv

Basic Transport Package

mt3d.btn

Dispersion Package

mt3d.dsp

Flow-transport link⁹²

modflow.linkmt3d

Generalized Conjugate Gradient Solver Package

mt3d.gcg

Lake Transport Package

mt3d.lkt

Name File

mt3d.nam

Reaction Package

mt3d.rct

Sink/Source Mixing Package

mt3d.ssm

Streamflow Transport Package

mt3d.sft

5.3.2.2. Output Files

Run Listing

mt3d.list

Concentration for the dissolved-phase

MT3Dnnn.UCN⁹³

Concentration for the sorbed-phase

MT3DnnnS.UCN⁹³

Mass budget summary

MT3Dnnn.MAS⁹³

Streamflow Transport Package Observation

mt3d.sft.obs

5.3.3. GSFLOW

5.3.3.1. Input Files

Control File

gsflow.control

Climate_Hru Files

 daily maximum temperature

gsflow.tmax.day

 daily minimum temperature

gsflow.tmin.day

 precipitation

gsflow.precip.day

 potential ET

gsflow.potet.day

 solar radiation

gsflow.swrad.day

Data File

gsflow.observationdata

Parameter Files

 Cascade Module

gsflow.cascade

 Dimensions

gsflow.dimensions

 Discretization

gsflow.discretization

⁹¹ Output files are optional and defined in the Options tab of the Subsidence and Aquifer Compaction dialog box (Figure 96)

⁹² This file is created by a MODFLOW run.

⁹³ “nnn” is the species index number, such as 001 for species 1, 002 for species 2, and so on.

Interception Module	gsflow.interception
Potential Evapotranspiration Module	gsflow.potentialevapotranspiration
Precipitation Distribution Module	gsflow.precipitation
Snow Computation Module	gsflow.snow
Soil Zone Module	gsflow.soilzone
Surface Runoff Module	gsflow.surfacerrunoff
Temperature Distribution Module	gsflow.temperature
Transpiration Period Module	gsflow.transpiration
MODFLOW Input Files	See Section 5.3.1.1 ⁹⁴

5.3.3.2. Output Files

Run Log File	gsflow.log
PRMS Water Budget File	gsflow.out.csv
HRU Summary Files	See Section 3.9.4.12
MODFLOW Output Files	See Section 5.3.1.2

5.3.4. SEAWAT

This section lists the input files specific to SEAWAT. In addition to these files, all input and output files of MODFLOW and MT3D-USGS may be included in a SEAWAT simulation run.

5.3.4.1. Input Files

Name File	seawat.nam
Variable Density Package	seawat.vdf
Viscosity Package	seawat.vsc

5.3.4.2. Output Files

Listing File	seawat.list
--------------	-------------

5.3.5. MODPATH

Most input and output files of a MODPATH instance use *MODPATH-InstanceID* as their base filename. The *InstanceID* is a string consists of 5-characters displayed on the Settings window of the MODPATH instance.

5.3.5.1. Input Files

Simulation	MODPATH-InstanceID.sim
Basic Data	MODPATH-InstanceID.bas
Discretization	modflow.dis
Flow (cell-by-cell flow terms) ⁹²	modflow.flow
Hydraulic head ⁹²	modflow.head
Name File	MODPATH-InstanceID.nam

⁹⁴ Not all MODFLOW packages are supported by GSFLOW for a coupled surface water and groundwater simulation. For details, refer to https://water.usgs.gov/water-resources/software/gflow/GSFLOW_Release_Notes_2.1.0.pdf

Starting Location

MODPATH-InstanceID.locations

5.3.5.2. Output Files

Pathline

MODPATH-InstanceID.pathline

Endpoint

MODPATH-InstanceID.endpoint

Time Series

MODPATH-InstanceID.timeseries

5.3.6. PEST

5.3.6.1. Input Files

Bore Coordinate

pest.borecoordinate

Control Data

pest.pst

Drawdown Instruction

pest.drawdowninstruction

Grid Specification

pest.gridspecification

Head Instruction

pest.headinstruction

Mod2SMP Input for extracting head results

pest.mod2smpheadinput

Mod2SMP Input for extracting drawdown results

pest.mod2smpdrawdowninput

Settings

settings.fig

Pilot Points⁹⁵

HP_Lnnn.PilotPoint

Template Files

Block-Centered Flow Package

pest.bcf6

Drain Package

pest.drn

Drain with Return Flow Package

pest.drt

Evapotranspiration Package

pest.evt

Evapotranspiration Segments Package

pest.ets

Flow and Head Boundary Package

pest.fhbm

General-Head Boundary Package

pest.ghb

Horizontal-Flow Barrier Package

pest.hfb6

Interbed-Storage Package

pest.ibs

Lake Package

pest.lak

Layer-Property Flow Package

pest.lpf

Pilot Points

MP_Lnnn.PilotPointTemplate⁹⁵

Recharge Package

pest.rch

Reservoir Package

pest.res1

River Package

pest.riv

Stream (STR) Package

pest.str

Streamflow-Routing (SFR2) Package

pest.sfr2

Unsaturated-Flow Package

pest.uzf

Upstream Weighting Package

pest.upw

⁹⁵ The string “MP” in the filenames will be replaced by the variable name of the respective hydraulic property. The string “nnn” will be replaced by the layer number where pilot points are located. For example, if pilot points represent the horizontal hydraulic conductivity values in layer 2, the name of a pilot point file is “HK_L2.PilotPoint”.

Well Package

pest.wel

5.3.6.2. Output Files

Run Record

pest.rec

Estimated Parameter Values

pest.par

Iteration Specific Parameter Values

pest.par.n⁹⁶

Run Specific Parameter Value

pest.par.n_m⁹⁶

Residuals

pest.res

Interim Residuals

pest.rei

Iteration Specific Residuals

pest.rei.n⁹⁶

Composite Parameter Sensitivities

pest.sen

Composite Observation Sensitivities

pest.seo

SVD Output

pest.svd

Matrix

pest.mtt

Resolution Data

pest.rsd

Jacobian Matrix

pest.jco

Iteration Specific Jacobian Matrix

pest.jco.n⁹⁶

Run Management Record

pest.rmr

5.3.6.3. Batch Files for starting PEST or BeoPEST

PEST Batch File

pest.bat

PESTCHEK Batch File

pestchek.bat

Model Run Batch File

moderun.bat

BeoPEST Master Batch File

BeoPEST.bat⁹⁷

BeoPEST Agent Batch File

BeoPEST_agent.bat⁹⁸

BeoPEST Agents Batch File

BeoPEST_agents.bat⁹⁹

5.3.7. PPK2FAC1

5.3.7.1. Input Files

Main Input

MP_Lnnn.PPK2FAC_InputFile⁹⁵

Grid Specification

pest.gridspecification

Pilot Points

MP_Lnnn.PilotPoint⁹⁵

Zone Array

MP_Lnnn.PilotPointZone⁹⁵

Structure

MP_Lnnn.Structure⁹⁵

5.3.7.2. Output Files

Kriging Factor

MP_Lnnn.KrigingFactor⁹⁵

Standard Deviation

MP_Lnnn.StdDeviation⁹⁵

Regularization Info

MP_Lnnn.RegularizationInfo⁹⁵

⁹⁶ See Control Data Tab of Section 3.9.5.2 for details.

⁹⁷ This file is created for users to start BeoPEST master manually.

⁹⁸ This batch file starts an instance of BeoPEST agent.

⁹⁹ This batch file starts all instances of BeoPEST agent.

5.3.7.3. Batch File for starting PPK2FAC1 and FAC2REAL

Kriging Batch File MP_Lnnn_Kriging.bat⁹⁵

5.3.8. FAC2REAL

5.3.8.1. Input Files

Main Input MP_Lnnn.FAC2REAL_InputFile⁹⁵
Kriging Factor MP_Lnnn.KrigingFactor⁹⁵
Pilot Points MP_Lnnn.PilotPoint⁹⁵

5.3.8.2. Output Files

Output Array MP_Lnnn.KrigingOutput⁹⁵

5.3.8.3. Batch File for starting PPK2FAC1 and FAC2REAL

Kriging Batch File MP_Lnnn_Kriging.bat⁹⁵

5.3.9. PPCOV

5.3.9.1. Input Files

Main Input MP_Lnnn.PPCOV_InputFile⁹⁵
Pilot Points MP_Lnnn.PilotPoint⁹⁵
Structure MP_Lnnn.Structure⁹⁵

5.3.9.2. Output Files

Covariance MP_Lnnn.Covariance⁹⁵

5.3.10. PPKREG1

5.3.10.1. Input Files

Main Input MP_Lnnn.PPKREG1_InputFile⁹⁵
Bore Coordinate pest.borecoordinate
PEST Control File pest.pst
Pilot Point MP_Lnnn.PilotPoint⁹⁵
Regularization Info MP_Lnnn.RegularizationInfo⁹⁵

5.3.10.2. Output Files

PEST Control File pest_temp.pst

References

- Alexander Martin** Biodegradation and Bioremediation [Book]. - Ithaca, New York : Academic Press, 1999. - Second Edition : p. 453. - ISBN-13: 978-0120498611.
- Anderson Mary P.** Groundwater Contamination [Report] : Consensus Study Report. - [s.l.] : The National Academies Press, 1984. - pp. 37-45. - ISBN13: 978-0-309-07832-0 .
- Anderson Mary P., Woessner William W. and Hunt Randall J.** Applied groundwater modeling [Book]. - [s.l.] : Elsevier, 2015. - 2 : p. 564. - ISBN 978-0-12-058103-0.
- Bakker Mark [et al.]** Documentation of the Seawater Intrusion (SWI2) Package for MODFLOW [Report] = Techniques and Methods 6–A46. - Reston, Virginia : U.S. Geological Survey, 2013.
- Banta Edward R.** MODFLOW-2000, The U.S. Geological Survey Modular Ground-Water Model— Documentation of Packages for Simulating Evapotranspiration with a Segmented Function (ETS1) and Drains with Return Flow (DRT1) [Report] = Open-File Report 00–466. - Denver, CO : U.S. Geological Survey, 2000.
- Banta Edward R.** Modifications to MODFLOW boundary conditions and an adaptive-damping scheme for Picard iterations for a highly nonlinear regional model [Conference] // MODFLOW and More 2006—Managing Ground Water Systems. - 2006. - pp. 596-600.
- Bear Jacob** Dynamics of fluids in porous media [Book]. - New York : American Elsevier Publicaiton Company,, 1972. - ISBN-13: 978-0486656755.
- Bear Jacob** Hydraulics of Groundwater [Book]. - New York : McGraw-Hill, 1979. - ISBN-13: 978-0486453552.
- Bedekar Vivek [et al.]** MT3D-USGS version 1.0.0: Groundwater Solute Transport Simulator for MODFLOW: U.S. Geological Survey Software Release, 30 September 2016 [Report]. - Reston, Virginia : [s.n.], 2016.
- Bedekar Vivek [et al.]** MT3D-USGS version 1: A U.S. Geological Survey release of MT3DMS updated with new and expanded transport capabilities for use with MODFLOW: U.S. Geological Survey Techniques and Methods 6–A53 [Report]. - Reston, Virginia : U.S. Geological Survey, 2016.
- Boyce S. E. [et al.]** One-Water Hydrologic Flow Model: A MODFLOW based conjunctive-use simulation software: U.S. Geological Survey Techniques and Methods 6–A60, 435 [Report]. - 2020.
- Carter J. R.** The effect of data precision on the calculation of slope and aspect using gridded DEMs [Journal]. - [s.l.] : Cartographica, 1992. - 1 : Vol. 29. - pp. 22-34.

Christiansen D. E. , Markstrom S. L. and Hay L. Impacts of climate change on growing season in the United States [Article] // Earth Interactions. - 2011. - 33 : Vol. 15. - pp. 1-17.

Defense Mapping Agency The Universal Grids: Universal Transverse Mercator (UM) and Universal Polar Stereographic (UPS). [Report] = DMATM 8358.2. - Fairfax, VA. : [s.n.], 1989.

Doherty John [Online]. - Watermark Numerical Computing and Groundwater Model Decision Support Initiative (GMDSI), December 2023.

Doherty John Calibration and uncertainty analysis for complex environmental models. [Book]. - Brisbane, Australia : Watermark Numerical Computing, 2015. - p. 227. - ISBN-13: 978-0-9943786-0-6.

Doherty John Groundwater Data Utilities [Report]. - [s.l.] : Watermark Numerical Computing, 2020.

Doherty John Model-Independent Parameter Estimation User Manual: 6th Edition. [Online]. - 2016.

Doherty John Model-Independent Parameter Estimation User Manual: 7th Edition. [Online]. - Water Numerical Computing, 2018.

Doherty John MODINV - Suite of software for MODFLOW pre-processing, postprocessing and parameter optimization. User's manual. [Report]. - Australia : Australian Centre for Tropical Research, 1990.

Domenico Patrick A. and Schwartz Franklin W. Physical and Chemical Hydrogeology [Book]. - [s.l.] : John Wiley & Sons, Inc., 1997. - 2nd. - ISBN-13: 978-0471597629.

Domenico Patrick A. Concepts and Models in Groundwater Hydrology [Book]. - [s.l.] : McGraw-Hill, 1972. - ISBN: 978-0070175358.

Dunne T. and Black R. G. An experimental investigation of runoff production in permeable soils [Article] // Water Resources Research. - 1970. - Vol. 6. - pp. 478-490.

ESRI ESRI Shapefile Technical Description -- An ESRI White Paper [Online] // ESRI.COM. - July 1998. - July 17, 2017. - <https://www.esri.com/library/whitepapers/pdfs/shapefile.pdf>.

Eyton J. R. Rate-of-change maps [Journal]. - [s.l.] : Cartography and Geographic Information Systems, 1991. - 2 : Vol. 18. - pp. 87-103.

Fenske Jon P., Leake Stanley A. and Prudic David E. Documentation of a computer program (RES1) to simulate leakage from reservoirs using the modular finite-difference ground-water flow model (MODFLOW) [Report] = Open-File Report 96-364. - Tucson, Arizona : U.S. Geological Survey, 1996.

Fetter Charles W. Applied Hydrogeology [Book]. - [s.l.] : Prentice-Hall, 1994. - 3rd Edition : p. 691. - ISBN: 978-0023364907.

Ford L. R. and Fulkerson D. R. Maximal flow through a network: Canadian Journal of Mathematics [Article]. - 1956. - pp. 399-404.

Freeze Allan R. and Cherry John A. Groundwater [Book]. - Upper Saddle River, NJ : Prentice-Hall, Inc., 1979. - ISBN 978-0133653120.

Gelhar Lynn W., Welty Claire and Rehfeldt Kenneth R. A critical review of data on field-scale dispersion in aquifers [Journal]. - [s.l.] : Water Resources Research, 1992. - 7 : Vol. 28.

Gesch D. [et al.] The National Elevation Dataset [Journal] // Photogrammetric Engineering & Remote Sensing. - [s.l.] : ASPRS, 2002. - 1 : Vol. 68.

Goodin William R., McRae Gregory J. and Seinfeld John H. A comparison of interpolation methods for sparse data: application to wind and concentration fields. [Journal] // Journal of Applied Meteorology and Climatology. - [s.l.] : American Meteorological Society, June 1979. - Vol. 18. - pp. 761-771.

Guo Weixing and Langevin Christian D. User's guide to SEAWAT; a computer program for simulation of three-dimensional variable-density ground-water flow [Report] = Techniques of Water-Resources Investigations 06-A7. - Tallahassee, Florida : U.S. Geological Survey, 2002.

Halford Keith J. and Hanson Randall T User Guide for the Drawdown-Limited, Multi-Node Well (MNW) Package for the U.S. Geological Survey's Modular Three-Dimensional Finite-Difference Ground-Water Flow Model, Versions MODFLOW-96 and MODFLOW-2000 [Report] = Open-File Report 02-293. - Sacramento, California : U. S. Geological Survey, 2002.

Hamon W. R. Estimating Potential evapotranspiration: Proceedings of the American Society of Civil Engineers, Journal of the Hydraulic Division, v. 87, no. HY3, p. 107-120. [Article]. - 1961.

Harbaugh Arlen W. [et al.] MODFLOW-2000, The U.S. Geological Survey modular ground-water model -- User guide to modularization concepts and the ground-water flow process. Open-File Report 00-92 [Report]. - Reston, Virginia : U.S. GEOLOGICAL SURVEY, 2000.

Harbaugh Arlen W. A Computer Program for Calculating Subregional Water Budgets Using Results from the U.S. Geological Survey Modular Three-dimensional Finite-difference Ground-water Flow Model. Open-File Report 90-392 [Report] = Open-File Report 90-392. - Reston, Virginia : U.S. Geological Survey, 1990. - p. 49.

Harbaugh Arlen W. and McDonald Michael G. User's documentation for MODFLOW-96, an update to the U.S. Geological Survey modular finite-difference ground-water flow model. Open-File Report 96-485 [Report]. - Reston, Virginia : U.S. GEOLOGICAL SURVEY, 1996.

Harbaugh Arlen W. Direct solution package based on alternating diagonal ordering for the U.S. Geological Survey modular finite-difference ground-water flow model [Report] = Open File Report 95-288. - Reston, Virginia : U.S. Geological Survey, 1995.

Harbaugh Arlen W. MODFLOW-2005, the U.S. Geological Survey modular ground-water model - the Ground-Water Flow Process: Techniques and Methods 6-A16. [Report]. - Reston, Virginia : U.S. Geological Survey, 2005.

Harbaugh Arlen W. ZONEBUDGET v.3.01, U.S. Geological Survey Software Release December 18, 2009.. - 2009.

Hargreaves G.H. and Samani Z. A. Estimating potential evapotranspiration. J. Irrig. and Drain Engr., ASCE, 108(IR3):223-230. [Article]. - 1982.

Hill Mary C. Preconditioned Conjugate-Gradient 2 (PCG2), A Computer Program for Solving Ground-Water Flow Equations [Report] = Water-Resources Investigations Report 90-4048. - Denver, Colorado : U.S. Geological Survey, 1990.

Hoffmann Jörn [et al.] MODFLOW-2000 Ground-water model--user guide to the Subsidence and Aquifer-System Compaction (SUB) Package [Report] = Open File Report 03-233. - Tucson, AZ : U.S. Geological Survey, 2003.

Horton R. E. The role of infiltration in the hydrological cycle [Article] // American Geophysical Union Transactions. - 1933. - Vol. 23. - pp. 479–482.

Hsieh Paul A. and Freckleton John R. Documentation of a computer program to simulate horizontal-flow barriers using the U.S. Geological Survey's modular three dimensional finite-difference ground-water flow model [Report] = Open-File Report 92–477. - Sacramento, California : U. S. Geological Survey, 1993.

Hunt Randall J. [et al.] Using the cloud to replenish parched groundwater modeling efforts. Rapid Communication for Ground Water, doi: 10.1111/j.1745-6584.2010.00699 [Journal] // Ground Water. - [s.l.] : Ground Water, 2010. - 3 : Vol. 48. - pp. 360-365.

Jensen M. E. and Haise H. R. Estimating Evapotranspiration from Solar Radiation [Article] // I. Irrig. Drain. Div., Am. Soc. Civil Eng. Proc. B9(IR4): 1 5-41 .. - 1963.

Konikow F. Leonard [et al.] Revised Multi-Node Well (MNW2) Package for MODFLOW Ground-Water Flow Model. Techniques and Methods 6-A30 [Report]. - [s.l.] : U.S. Geological Survey, 2009.

Langevin C.D., Thorne D.T., Jr. and Dausman A.M. SEAWAT Version 4: A Computer Program for Simulation of Multi-Species Solute and Heat Transport: U.S. Geological Survey Techniques and Methods Book 6, Chapter A22, 39 p. [Report]. - Reston, Virginia : U.S. Geological Survey, 2007.

Leake Stanley A. and Prudic David E. Documentation of a computer program to simulate aquifer-system compaction using the modular finite-difference ground-water flow model [Report] = Techniques of Water-Resources Investigations 06-A2. - [s.l.] : U. S. Geological Survey, 1991.

Leake Stanley. A. and Lilly Michael R. Documentation of a Computer Program (FHB1) for Assignment of Transient Specified-Flow and Specified-Head Boundaries in Applications of the Modular Finite-Difference Ground-Water Flow Model (MODFLOW) [Report] = Open-File Report 97-571. - Tucson, Arizona : U. S. Geological Survey, 1997.

Leavesley G. H. [et al.] Precipitation-runoff modeling system—User’s manual: U.S. Geological Survey Water-Resources Investigations Report 83–4238, 207 p. [Report]. - 1983.

Leavesley G. H. [et al.] The Modular Modeling System (MMS)—The physical process modeling component of a database-centered decision support system for water and power management [Article] // Water, Air, and Soil Pollution. - 1996. - Vol. 90.

Leonard B. P. A stable and accurate convective modeling procedure based on quadratic upstream interpolation. [Journal]. - [s.l.] : Computer Methods in Applied Mechanics and Engineering, 1979. - Vol. 19. - pp. 59-98.

Leonard B. P. Universal Limiter for transient interpolation modeling of the advective transport equations: the ULTIMATE conservative difference scheme [Report] = NASA Technical Memorandum 100916 ICOMP-88-11 / Institute for Computational Mechanics in Propulsion Lewis Research Center. - Cleveland, Ohio : NASA, 1988.

MapQuest Open Elevation API. - [s.l.] : MapQuest, Inc., 2019.

Markstrom S. L. [et al.] PRMS-IV, the precipitation-runoff modeling system, version 4: U.S. Geological Survey Techniques and Methods, book 6, chap. B7, 158 p. [Report]. - 2015.

Markstrom Steven G. [et al.] GSFLOW - Coupled Ground-Water and Surface-Water Flow Model Based on the Integration of the Precipitation-Runoff Modeling System (PRMS) and the Modular Ground-Water Flow Model (MODFLOW-2005) [Report] = Techniques and Methods 6-D1. - [s.l.] : U. S. Geological Survey, 2008.

McDonald Michael G. and Harbaugh Arlen W. MODFLOW, A modular three-dimensional finite difference ground-water flow model [Report] = TWRI-6-A1. - Reston, Virginia : U. S. Geological Survey, 1988.

McDonald Michael G. [et al.] A method of converting no-flow cells to variable-head cells for the U. S. Geological Survey modular finite-difference ground-water flow model [Report] = Open-File Report 91-536. - [s.l.] : U. S. Geological Survey, 1991.

Mehl Steffen W. and Mary Hill C. MODFLOW-2000, the U. S. Geological Survey modular ground-water model; user guide to the Link-AMG (LMG) package for solving matrix equations using an

algebraic multigrid solver [Report] = Open-File Report 2001-177. - Denver, Colorado : U. S. Geological Survey, 2001.

Merritt Michael L. and Konikow Leonard F. Documentation of a computer program to simulation lake-aquifer interaction using the MODFLOW groundwater flow model and the MOC3D solute-transport model [Report] = Water-Resources Investigation Report 00-4167 / U.S. Geological Survey. - Tallahassee, FL : [s.n.], 2000. - p. 146.

Monteith J. L. Evaporation and the Environment. 19th Symposia of the Society for Experimental Biology, 19, 205-234. [Article]. - 1965.

Mount David and Arya Sunil ANN: Approximate Nearest Neighbors. - College Park : Dept of Computer Science, University of Maryland, 2010.

Naff Richard L. and Banta Edward R. The U.S. Geological Survey Modular Ground-Water Model - PCGN: A Preconditioned Conjugate Gradient Solver with Improved Nonlinear Control [Report] = Open-File Report 2008-1331. - Reston, Virginia : U. S. Geological Survey, 2008.

National Imagery and Mapping Agency Department of Defense World Geodetic System 1984. Its Definition and Relationships with Local Geodetic Systems. [Report] = NIMA TR8350.2. Third Edition. - 1984.

Neville Christopher J. Interpreting the MT3D Mass Budget [Online] // www.sspa.com. - S.S. Papadopoulos, Inc., June 7, 2006. - February 19, 2018. - http://www.sspa.com/sites/default/files/images/stories/documents/MT3D-notes_mass-budget.pdf.

Niswonger R. G., Prudic D. E. and Regan R. S. Documentation of the Unsaturated-Zone Flow (UZF1) Package for modeling unsaturated flow between the land surface and the water table with MODFLOW-2005: U.S. Geological Techniques and Methods Book 6, Chapter A19 [Report]. - 2006a.

Niswonger Richard G., Panday Sorab and Ibaraki Motomu MODFLOW-NWT, A Newton formulation for MODFLOW-2005: U.S. Geological Survey Techniques and Methods 6-A37, 44 p. [Report]. - Reston, Virginia : U.S. Geological Survey, 2011.

Niswonger Richard G. and Prudic David E. Documentation of the Streamflow-Routing (SFR2) Package to Include Unsaturated Flow Beneath Streams—A Modification to SFR1 [Report] = Techniques and Methods 6-A13. - [s.l.] : U.S. Geological Survey, 2006.

Niswonger Richard G., Prudic David E. and Regan Steven R. Documentation of the Unsaturated-Zone Flow (UZF1) Package for Modeling Unsaturated Flow Between the Land Surface and the Water Table with MODFLOW-2005 [Report] = Techniques and Methods 6-A19. - Reston, Virginia : [s.n.], 2006.

Peaceman Donald W. Interpretation of well-block pressures in numerical reservoir simulation with nonsquare grid blocks and anisotropic permeability [Journal]. - [s.l.] : Society of Petroleum Engineers Journal, June 1983. - 3 : Vol. 23. - pp. 531–543.

Penman H. L. Natural Evaporation from Open Water, Bare Soil and Grass. Proceedings of the Royal Society of London, 193, 120-145. [Article]. - 1948.

Pollock David W. User guide for MODPATH Version 7 -- A particle-tracking model for MODFLOW: U.S. Geological Survey Open-File Report 2016-1086, 35 p., <http://dx.doi.org/10.3133/ofr20161086>. [Report]. - Reston, Virginia : U.S. Geological Survey, 2016.

Priestley C. H. B. and Taylor R. J. On the assessment of surface heat flux and evaporation using large-scale parameters. Mon. Weather Rev. 100: 81– 92 [Article]. - 1972.

Prudic David E. Documentation of a computer program to simulate stream-aquifer relations using a modular, finite-difference, ground-water flow model [Report] = Open-File Report 88-729. - Carson City, Nevada : U. S. Geological Survey, 1989.

Prudic David E., Konikow Leonard F. and Banta Edward R. A New Streamflow-Routing (SFR1) Package to Simulate Stream-Aquifer Interaction with MODFLOW-2000 [Report] : Open-File Report. - [s.l.] : U.S. Geological Survey, 2004.

Regan R. S. and LaFontaine J. H. Documentation of the Dynamic Parameter, Water-Use, Stream and Lake Flow Routing, and Two Summary Output Modules and Updates to Surface-Depression Storage Simulation and Initial Conditions Specification Options [Report] = Techniques and Methods 6–B8. - [s.l.] : U. S. Geological Survey, 2017.

Regan R. S. [et al.] GSFLOW version 1.2.2: Coupled Groundwater and Surface-water FLOW model: U.S. Geological Survey Software Release, 23 February 2018. [Report]. - 2018.

Regan R. S. and Niswonger R. G. GSFLOW version 2.1.0: Coupled Groundwater and Surface-water FLOW model: U.S. Geological Survey Software Release, 04 March 2020.. - 2020.

Schwartz Joe Bing Maps Tile System [Online] // MSDN Library. - 6 8, 2022. - September 26, 2023. - <https://learn.microsoft.com/en-us/bingmaps/articles/bing-maps-tile-system?redirectedfrom=MSDN>.

Spitz Karlheinz and Moreno Joanna A Practical Guide to Groundwater and Solute Transport Modeling [Book]. - [s.l.] : Wiley, 1996. - ISBN: 978-0-471-13687-3.

Tangborn W.V. A model to predict short-term snowmelt runoff using synoptic observations of streamflow, temperature, and precipitation, in Modeling of Snow Cover Runoff [Report]. - [s.l.] : U.S. Army Corps of Engineers, Cold Region Research Laboratory,, 1978.

Thiem Günter Hydrologische Methoden [Book]. - Leipzig, Germany : J.M. Gebhart, 1906. - p. 56.

Thompson E. S. Computation of solar radiation from sky cover [Article] // Water Resources Research. - 1976. - 5 : Vol. 12. - pp. 859–865.

Watermark Numerical Computing Groundwater Data Utilities [Online]. - 2020. - March 2022. - <https://s3.amazonaws.com/docs.pesthomepage.org/software/gwutils.zip>.

Welter David E. [et al.] Approaches in Highly Parameterized Inversion: PEST++ Version 3, A Parameter ESTimation and Uncertainty Analysis Software Suite Optimized for Large Environmental Models [Article]. - [s.l.] : U.S. Geological Survey, 2015. - Vols. Techniques and Methods 7–C12.

White Jeremy T. [et al.] Approaches to Highly Parameterized Inversion: PEST++ Version 5, a Software Suite for Parameter Estimation, Uncertainty Analysis, Management Optimization and Sensitivity Analysis [Report]. - [s.l.] : U. S. Geological Survey, 2020.

White Jeremy T. A model-independent iterative ensemble smoother for efficient history-matching and uncertainty quantification in very high dimensions. [Article] // Environmental Modelling and Software. - [s.l.] : Elsevier, 2018. - Vol. 109.

Wilson John D. and Naff Richard L. MODFLOW–2000, The U.S. Geological Survey Modular Ground–Water Model – GMG Linear Equation Solver Package Documentation [Report] = Open-File Report 2004-1261 / U. S. Geological Survey. - Denver, Colorado : [s.n.], 2004.

Zheng Chunmiao and Bennett Gordon D. Applied contaminant transport modeling: Theory and practice [Book]. - New York : Van Nostrand Reinhold, 1995. - 1st : p. 440. - ISBN-13: 9780442013486.

Zheng Chunmiao and Wang Patrick P. MT3DMS: A modular three-dimensional multispecies Transport Model for Simulation of Advection, Dispersion, and Chemical Reactions of Contaminants in Groundwater Systems; Documentation and User’s Guide, Contract Report SERDP-99-1 [Report]. - Vicksburg, MS : U.S. Army Engineer Research and Development Center, 1999.

Index

2

2D Matrix85, 333
2D Modflow Array 81, 82, 85, 333
2D Weight Matrix.....333

3

3D Modflow Array 81, 85, 334

A

active cell29
animation.....57
anisotropy
 horizontal.....95
 vertical96
ASCII Raster..... 81, 85

B

BeoPEST2, 61
bulk density.....98

C

Cartesian coordinate system50
cascading-flow241
climate_hru.....245
contours
 input parameter311
 model results317
Copy Cell Data74, 78
Copy Grid Size78
Copy Grid Split Factor78
Copy Layer Data75, 78
Cross section74, 76
CSV..... ix, 209, 335

D

Data
 export 85
 import..... 81, 84, 85
 modify 79
diffusion coefficient 226

E

effective porosity 97
Export..... 85

F

First-order Euler 224
flow vectors..... 321
Fourth-order Runge-Kutta..... 224

G

geographic coordinate system 50
georeferencing method..... 54
grid azimuth 54
grid cursor 76
grid editor..... 25, 27
ground surface elevation 4
Groundwater Flow Package 9
GSFLOW..... 2, 60

H

HMOC..... 224
HRU 236
hydraulic conductivity
 horizontal 95
 vertical..... 96
Hydrologic Response Units..... 236

I

IBOUND.....	94
ICBUND.....	94
import.....	81
Import Data.....	81, 84, 85
Import Package	86, 87
input method	
cell-by-cell.....	6, 29
pilot points.....	40
polygon.....	6
polyline.....	6
interpolation.....	83

K

Kriging	
apply kriging.....	48

L

layer type.....	90
longitudinal dispersivity.....	226
LSQR.....	284

M

MMOC.....	223
MOC.....	223
model	
create.....	52
import.....	54
model data folder.....	52
model grid	
move.....	65
rotate.....	28, 65
MODFLOW Version.....	9
MODFLOW-2005.....	59
MODFLOW-NWT.....	2, 9, 60, 190, 221
MODFLOW-OWHM.....	2, 60
Modify Data.....	79
MODPATH.....	61, 325
MT3DMS.....	1, 60, 221
MT3D-USGS.....	2, 60, 209

N

National Elevation Dataset.....	5
---------------------------------	---

O

observation data.....	66
observation points.....	66

P

Package	
Advection (ADV).....	222
Direct Solver (DE4).....	177
Dispersion (DSP).....	226
Drain (DRN).....	98
Drain with Return Flow (DRT1).....	101
Evapotranspiration (EVT).....	104
Evapotranspiration Segments (ETS1).....	105
Flow and Head Boundary (FHB1).....	107
General-Head Boundary (GHB).....	111
Horizontal-Flow Barrier (HFB).....	114
import.....	87
Interbed Storage (IBS).....	115
Lake (LAK3).....	117
Multi-Node Well (MNW1).....	121
Multi-Node Well 2(MNW2).....	124
Newton Solver (NWT).....	189
PCG with Improved Nonlinear Control (PCGN).....	185
Preconditioned Conjugate Gradient Solver (PCG).....	183
Reaction (RCT).....	226
Recharge (RCH).....	133
Reservoir (RES1).....	137
River (RIV).....	134
Seawater Intrusion 2 (SWI2).....	164
Sink/Source Mixing (SSM).....	227
Stream (STR).....	140
Streamflow-Routing (SFR2).....	146
Subsidence and Aquifer System Compaction (SUB).....	160
Time-Variant Specified-Head (CHD).....	168
Unsaturated-Zone Flow (UZF1).....	169
Variable-Density Flow (VDF).....	228
Viscosity (VSC).....	228
Well (WEL).....	172
parameter group.....	275
particle tracking.....	325
add particles.....	326
add starting points.....	326
delete particles.....	328
delete starting points.....	328
particles.....	326
add.....	326
delete.....	328
particle-tracking algorithm.....	224
pathlines.....	332
PEST.....	61, 264, 291

PEST Settings.....	271, 304
PESTCHEK.....	61
PESTPP-GLM.....	61
control variables.....	290
PESTPP-IES.....	61, 305
control variables.....	290
Pilot point	
add a group of pilot points.....	42
delete.....	43
edit data	44
import from pilot point files.....	43
input method.....	40
Potential Evapotranspiration Modules.....	245
Precipitation Distribution Module.....	245
preconditioned conjugate gradient.....	183, 185
preferences.....	59
prior information.....	277
prj19, 57	
PRMS Process	
Interception.....	242
Potential Evapotranspiration.....	243
Precipitation distribution.....	245
Snow computation.....	249
Soil zone.....	250
Solar radiation.....	252
Surface runoff.....	253
Temperature distribution.....	256
Transpiration period.....	260
PRMS Processes.....	242
Proxy.....	63

Q

quadkey.....	17
--------------	----

R

regularization mode.....	272
release time.....	326
retardation factor.....	97
Run	
MODFLOW.....	195, 263
MODPATH.....	329
MT3D.....	231
PEST.....	293
PESTCHEK.....	306
SEAWAT.....	231
ZoneBudget.....	208

S

sandbox.....	98
SEAWAT.....	1, 60, 209, 218
seawater intrusion.....	164
sensitivity analysis.....	271
Settings window	
of a model.....	7
Shapefile.....	ix, 19, 81
Simulation Length.....	89
Simulation Time.....	89
Solvers	
MODFLOW.....	177
MT3D.....	228
PEST.....	272
specific storage.....	96
specific yield.....	97
starting hydraulic head.....	95
starting points.....	See particles
storage coefficient.....	96
supported models.....	2
paths to the executable files.....	59
SURFER GRD.....	81, 85
SVD.....	283

T

table of contents.....	3
temporal discretization.....	25, 88
tiled-map services.....	17
time unit.....	88
transmissivity.....	95

U

ULTIMATE.....	222
units.....	25

V

Variogram.....	45
----------------	----

W

Wetting Threshold.....	175
World file.....	57
World Geodetic System 1984.....	ix

Z

Zone Budget..... 2, 62, 208

