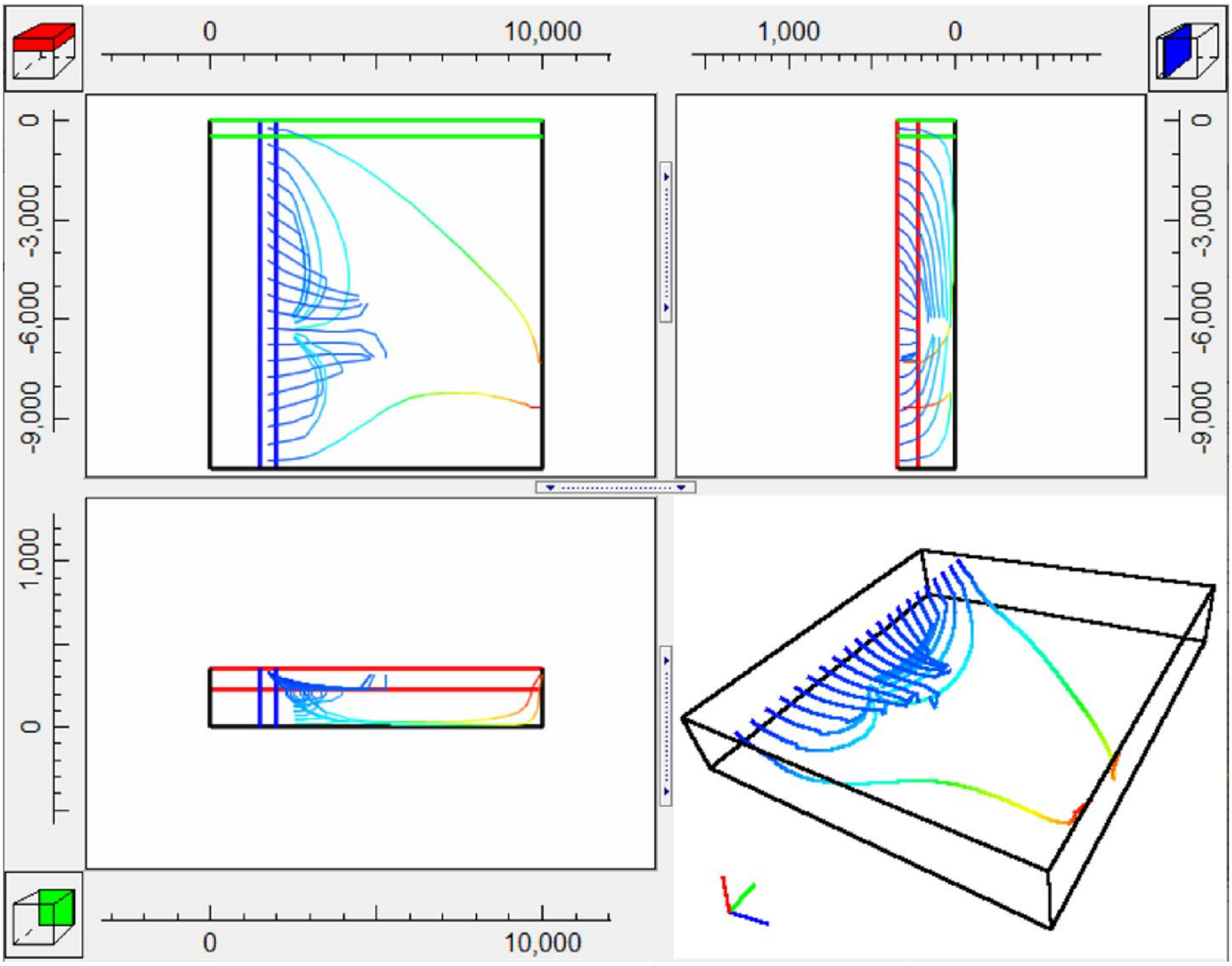


Getting Started with MODFLOW

Richard B. Winston

THE GROUNDWATER PROJECT

Getting Started with MODFLOW

The Groundwater Project

The Groundwater Project relies on private funding for book production and management of the Project.

Please consider making a donation to the Groundwater Project↗ so books will continue to be freely available.

Thank you.

Richard B. Winston
College Park, Maryland, USA

Getting Started with MODFLOW

The Groundwater Project
Guelph, Ontario, Canada
Version 6, February, 2024

Published by the Groundwater Project, Guelph, Ontario, Canada, 2023.

Winston, Richard B.

Getting Started with MODFLOW / Richard B. Winston – Guelph, Ontario, Canada, 2023.

242 pages

ISBN: 978-1-77470-030-3

DOI: https://doi.org/10.21083/978-1-77470-030-3↗

Please consider signing up to the Groundwater Project mailing list and stay informed about new book releases, events, and ways to participate in the Groundwater Project. When you sign up to our email list, it helps us build a global groundwater community. Sign-up↗.

APA 7th Edition Citation:

Winston, Richard B. (2023). *Getting Started with MODFLOW*↗. The Groundwater Project, Guelph, Ontario, Canada. https://doi.org/10.21083/978-1-77470-030-3↗.

THE GROUNDWATER PROJECT

Cover Image: Richard B. Winston, 2022

Table of Contents

The Groundwater Project Foreword

At the United Nations (UN) Water Summit held on December 2022, delegates agreed that statements from all major groundwater-related events will be unified in 2023 into one comprehensive groundwater message. This message will be released at the UN 2023 Water Conference, a landmark event that will bring attention at the highest international level to the importance of groundwater for the future of humanity and ecosystems. This message will bring clarity to groundwater issues to advance understanding globally of the challenges faced and actions needed to resolve the world's groundwater problems. Groundwater education is key.

The 2023 World Water Day theme *Accelerating Change* is in sync with the goal of the Groundwater Project (GW-Project). The GW-Project is a registered Canadian charity founded in 2018 and committed to the advancement of groundwater education as a means to accelerate action related to our essential groundwater resources. To this end, we create and disseminate knowledge through a unique approach: the democratization of groundwater knowledge. We act on this principle through our website gw-project.org/↗, a global platform, based on the principle that

"Knowledge should be free, and the best knowledge should be free knowledge." Anonymous

The mission of the GW-Project is to promote groundwater learning across the globe. This is accomplished by providing accessible, engaging, and high-quality educational materials—free-of-charge online and in many languages—to all who want to learn about groundwater. In short, the GW-Project provides essential knowledge and tools needed to develop groundwater sustainably for the future of humanity and ecosystems. This new type of global educational endeavor is made possible through the contributions of a dedicated international group of volunteer professionals from diverse disciplines. Academics, consultants, and retirees contribute by writing and/or reviewing the books aimed at diverse levels of readers from children to high school, undergraduate and graduate students, or professionals in the groundwater field. More than 1,000 dedicated volunteers from 127 countries and six continents are involved—and participation is growing.

Hundreds of books will be published online over the coming years, first in English and then in other languages. An important tenet of GW-Project books is a strong emphasis on visualization; with clear illustrations to stimulate spatial and critical thinking. In future, the publications will also include videos and other dynamic learning tools. Revised editions of the books are published from time to time. Users are invited to propose revisions.

We thank you for being part of the GW-Project Community. We hope to hear from you about your experience with the project materials, and welcome ideas and volunteers!

The GW-Project Board of Directors

January 2023

Foreword

Numerous reports, journal publications, and a few textbooks have been written about groundwater modeling. Although these resources provide vital information for the wise construction and use of models, they do not explain how to get started. The Groundwater Project is pleased to publish this book, *Getting Started with MODFLOW*, to fill that gap; its readers will learn, step-by-step, how to build their first groundwater model.

Numerical computer codes for the simulation of groundwater flow in one- and two-dimensions first appeared in the 1960s but did not gain substantial usefulness until the late 1970s when computer access became affordable, rendering numerical groundwater modeling feasible for practical application. At the same time, modeling software advanced to include three-dimensional capabilities, giving groundwater scientists the freedom to simulate groundwater flow without the restrictions associated with analytical models. Early on, the US Geological Survey (USGS) was a leader in development and application of numerical models for groundwater applications which led to the first publicly available version of the MODFLOW code in 1984. The USGS re-issues MODFLOW every few years with expanded capabilities and greater ease of use while keeping it in the public domain.

This unique book goes far beyond user manuals and modeling texts to explain the use of MODFLOW in detail, including installation of the codes and step-by-step exercises that the reader can transfer to their own field sites. In particular, the final exercise demonstrates how the tools described throughout the book are put to work using data from a field site.

Groundwater flow simulation is a powerful tool that is useful even on a personal computer. However, it is not routinely used to assess and improve the conceptual models that underpin our understanding of flow-systems. Using MODFLOW, the modeler can gain insight into the hydrogeologic controls on flow in the groundwater system using simple exploratory models as thinking tools. Further, they can design a data collection program that will improve the underlying conceptual model. In short, MODFLOW should be used in all phases of a project, not only in the final phase. *Getting Started with MODFLOW* facilitates use of modeling in early phases by making modeling accessible to hydrogeologists who do not specialize in modeling.

Richard Winston has been instrumental in providing hands-on, how-to, educational materials and tools to the groundwater modeling community that range from a multitude of multimedia items↗ to extensive software tools for practical model application↗. His experience with preparing these materials has culminated in this effective, pragmatic guide that will enable those with a working knowledge of groundwater flow to become a MODFLOW user so that flow modeling becomes accessible outside of the domain of the modeling specialist.

John Cherry, The Groundwater Project Leader

Guelph, Ontario, Canada, November 2023

Preface

For over twenty years, part of my job at the U.S. Geological Survey (USGS) has been to provide support for groundwater modelers. I do this in several ways by:

- developing graphical user interfaces to help modelers create, run, and view the results of their models,
- maintaining the Online Guide to MODFLOW which describes the formats of the MODFLOW-2005 input files, and
- answering questions from modelers.

I answer a lot of questions, for example in 2022, I responded to over 400 inquiries. Some of these were simple; some not so simple. When I first started to support groundwater modeling, the questions were mostly about how to correctly format input files. That is no longer the case. An extremely small number of people ask about input formatting now because the various graphical user interfaces for MODFLOW including the Python package FloPy make it easy for people to create correctly formatted files. Now, many of the questions are from beginning modelers who need to understand how to use MODFLOW. This book is meant to address that problem. It is aimed at enabling beginners to use MODFLOW successfully. It won't make the reader an expert; that requires more study and experience. It may, however, start them on the road to being a successful MODFLOW modeler.

Acknowledgments

I deeply appreciate the thorough and useful reviews of and contributions to this book by the following individuals:

- Eve L. Kuniansky, Emeritus Scientist, United States Geological Survey, USA;
- Eileen Poeter, Professor Emeritus, Colorado School of Mines, USA;
- Claire R. Tiedeman, Retired, Sunnyvale, California, USA;
- Paul Hsieh, Retired, Redwood City, California, USA.

I am grateful for Amanda Sills of the Groundwater Project for their oversight and copyediting of this book. I thank Eileen Poeter (Colorado School of Mines, Golden, Colorado, USA) for producing this book.

1 Introduction

Every groundwater model starts with a problem. For example, if a farmer applies for a permit for a new irrigation well, the regulator may wish to know whether pumping from the well will cause other nearby wells to go dry. The regulator may use a model to predict the effects of the well on other groundwater users. From those results, the regulator may choose to approve or not approve the proposed well. Another example would be a city that wants to protect its water supply from pollution. It might use a model to help determine whether its water supply is in danger and if so, what to do about it. MODFLOW is often used to solve such problems either alone or in combination with other programs.

MODFLOW is a popular program for modeling groundwater flow. It was developed by the United States Geological Survey (USGS). It was first released in 1984 with later major releases in 1988, 1996, 2000, 2005, and 2017 (McDonald and Harbaugh, 1984, 1988, Harbaugh and McDonald, 1996a, 1996b, Harbaugh et al., 2000, Hill et al., 2000, Harbaugh, 2005, Langevin et al., 2017 Hughes et al., 2017, Provost et al., 2017).

The goal of this book is to teach enough about MODFLOW for the reader to be able to create a reasonable groundwater flow model of a study site and understand what the results mean. This book is designed to present enough about MODFLOW for the reader to be able to start using it. This book does not describe how to get the data needed for the MODFLOW input files. Instead, it focuses on providing a conceptual understanding of how to use MODFLOW. While it does not entirely ignore the mathematics behind MODFLOW, the mathematics are not the primary focus of this book. In this book, it is assumed that the reader will be using a graphical user interface to create the input files for MODFLOW, so the precise structure of the input files is largely neglected.

This book introduces the process of modeling groundwater flow using the most recent version of MODFLOW: MODFLOW 6. It assumes the reader knows the basics of groundwater flow. For example, it assumes Darcy's Law, hydraulic conductivity, specific storage, and specific yield are understood. A reader who does not already know the basics of groundwater flow should learn them before trying to learn MODFLOW. Several other books in the Groundwater Project can help with that. For example, Hydrogeologic Properties of Earth Materials and Principles of Groundwater Flow↗ introduces basic concepts such as Darcy's Law and describes hydraulic parameters. Other related books include Geologic Frameworks for Groundwater Flow Models↗, which describes the process of building the geologic framework of a model and Groundwater Resource Development: Effects and Sustainability↗, which discusses how to use MODFLOW to assess the effects of groundwater resource development on baseflow in streams.

Two other publications from the USGS are especially helpful. Reilly (2001)↗ and Reilly and Harbaugh (2004)↗ provide guidance on how to best represent the physical features and properties that control groundwater flow in a mathematical model such as MODFLOW.

1.1 Assumptions in MODFLOW

MODFLOW is primarily used for simulating laminar, constant-density flow of a single fluid in the saturated zone. It can simulate confined flow, unconfined flow, steady-state flow, and transient flow. MODFLOW models can be one-, two-, or three-dimensional. MODFLOW is not suitable for some problems because of the assumptions embedded in it. It is not suitable for modeling more than a single fluid. For example, it cannot model gasoline leaked from a gas station together with groundwater when the gasoline is a separate liquid phase. It is not suitable for modeling groundwater that varies significantly in density such as a problem involving salt-water mixing with fresh water (unless the Buoyancy package is used). Large temperature differences within the model can also be a problem if they cause the density of the water to vary enough to impact the hydraulic gradient. Differences in temperature and solute concentration can affect the viscosity of water and thus the hydraulic conductivity. This effect can now be simulated by MODFLOW. MODFLOW 6 cannot model turbulent flow such as might occur in caves. However, there is a variant of MODFLOW called MODFLOW-CFP that can model turbulent flow (Shoemaker et al., 2007). For situations in which MODFLOW is not suitable, there are a variety of analytical, analytical element, finite-difference, and finite-element groundwater modeling programs that can be used.

MODFLOW simulates groundwater flow by first discretizing the aquifer volume to be modeled into a grid of cells and then solving a system of equations at each active cell in the model. The equations represent the flow between the cell and its neighbors, flow into or out of the groundwater system, and changes in the amount of water stored in the cell. To calculate the flows, MODFLOW needs the heads in all the cells. However, the heads in the cells are also unknown and need to be calculated. To get around this, it uses the most recent known heads. For the first time step in the model, these are the initial heads provided by the modeler. For each succeeding time step, these will be the heads from the previous time step. Using these heads, MODFLOW calculates new values for the flows and heads for the current time step. Then, instead of advancing to the next time step, it repeats the calculations using the newly calculated heads. It repeats this process, iterating until the heads and flows calculated in an iteration are sufficiently similar to those in the previous iteration based on a tolerance provided by the modeler. At that point, the model has reached the convergence threshold, so those results are accepted as the solution for that time step and the model is said to have converged for that time step. If the model is poorly constructed or there are numerical problems, MODFLOW may not be able to find an acceptable solution and will halt with an error message. More detail about MODFLOW's computational process is provided in Box 1↴.

Often getting the first time-step to converge for a transient or steady-state model can be difficult and may require adjusting the time steps, and/or modifying solver parameters. Commonly, the first stress period of a transient model is a steady-state stress period that will provide heads consistent with the model properties and boundary conditions so the ensuing transient simulation represents changes in the system due to the stresses input by the modeler without adjustments for inconsistencies of the initial heads.

1.2 First Steps

To use MODFLOW, you will need to define the purpose of the model and specify the following model inputs.

- extent of the area to include in the model;
- discretization of the model area (the grid);
- hydraulic properties of the geologic units included in the model (hydraulic conductivity, and for transient simulation specific storage, and specific yield);
- initial heads at every active cell in the model;
- locations and properties of boundary conditions such as streams, drains, or springs; and
- locations and rates of recharge, well discharge, and evapotranspiration.

Often, one of the most important of these, possibly the most important, is recharge. Recharge is the water that enters the groundwater system by percolation through the material overlying the saturated zone. Typically, recharge comes from precipitation, but it can also come from losing streams, infiltration basins, or other sources. Not all precipitation contributes to recharge; some of it is evaporated or transpired before reaching the saturated zone. Recharge is important because it often is the single greatest driver of flow in the saturated zone.

Usually, there are many, and often large, uncertainties in the model inputs. Comparisons of data observed in the groundwater system and equivalent values simulated by MODFLOW are used to adjust the model inputs with the objective of better representing the groundwater flow system. This process is called calibration or calibrating the model.

Several types of data can be used to calibrate a model. Heads are the simplest type of observation to make and can be informative. However, heads should not be the only type of observation used to calibrate a model because using only heads can result in a wide range of parameter values providing an equally acceptable fit to the observations. Flow observations are especially important because they can help in estimating recharge rates and determining the absolute (as opposed to relative) values of hydraulic conductivity. Flow observations include groundwater discharge to surface water features such as a river, as well as withdrawal/injection rates from/to wells. Typically, it is easier to measure the water leaving the groundwater system as discharge to surface water bodies or wells than the water entering the system as recharge because generally discharge occurs over a focused area while recharge is distributed over a broad area. The use of climatic water budgets, evapotranspiration, and streamflow data all help modelers constrain simulated flows to or from the groundwater system and thus provide a more representative calibrated model. The numerical model calibration process is akin to using flow rate and drawdown data from an aquifer pumping test to estimate aquifer properties.

One mistake frequently made by beginners is to wait to start modeling until after collecting all the data for the model. Instead, modeling should begin as soon as the purpose of the model has been defined and the extent of the area to include in the model has been determined. Results from these early models are not expected to be realistic; however, these early models help in two ways. They can provide insight into the controls on flow in the system and they can help the modeler to design a data collection program that will be useful to improving the model. Because this modeling effort begins with incomplete data, the missing data values need to be estimated. These preliminary models need to be good enough to run to completion with an acceptably low water budget error. However, it would be a mistake to spend a large amount of time calibrating the model at this stage. Instead, the modeler strives to understand what the largest discrepancies between the model results and measured observations reveal about the model parameters. For example, if the heads are higher than the land surface over large parts of the model that might mean recharge rates are too high or hydraulic conductivities are too low. Then more effort could be directed toward improving estimates of recharge rates, or to conducting aquifer tests to estimate hydraulic conductivities, or both (Figure 1).

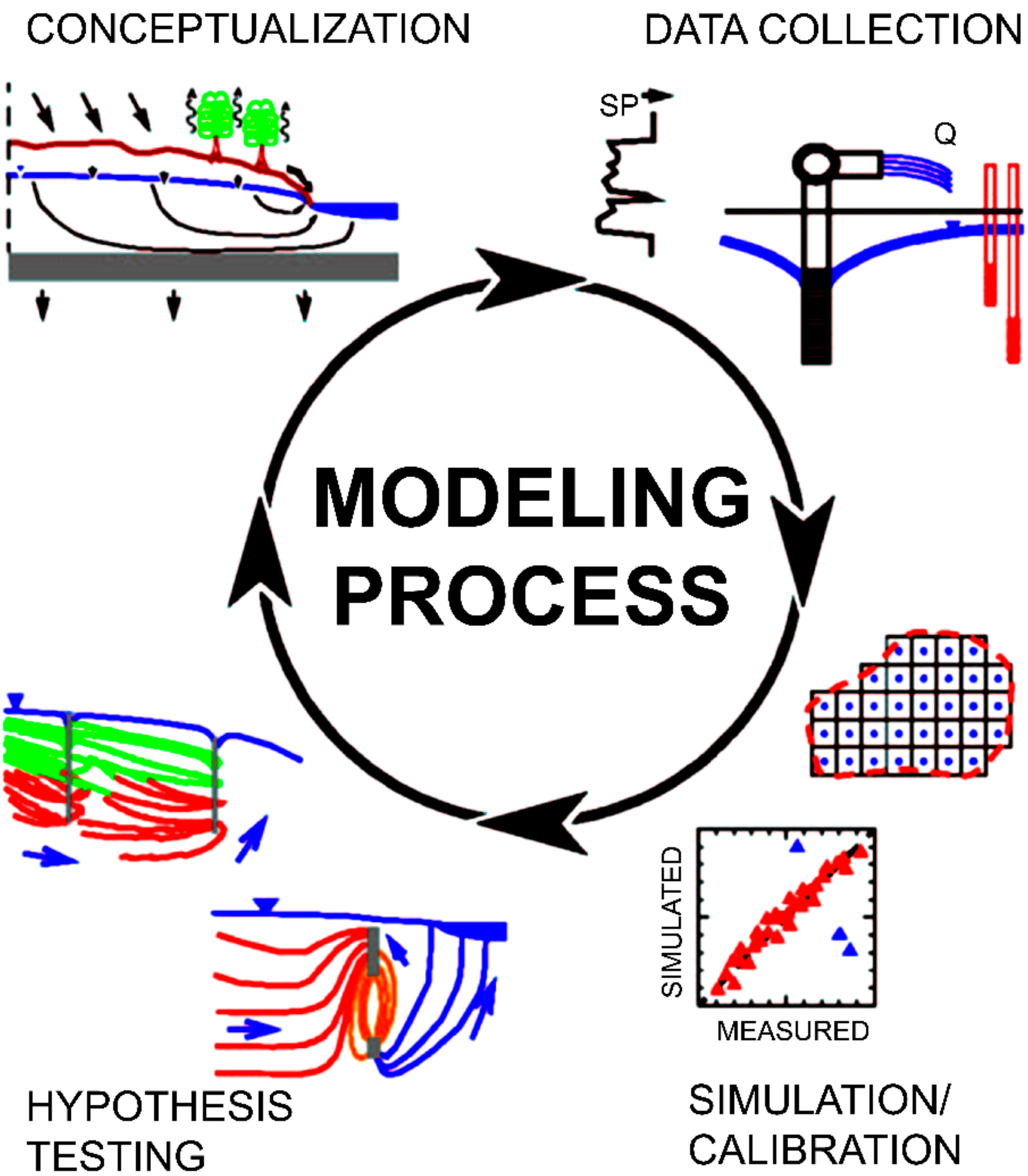


Figure 1 - The circular modeling process. First develop a conceptual model and compile existing data about the aquifer. Then collect or compile more data if required for the problem at hand. Decide on a modeling approach and calibration strategy, then test your hypothesis. If the model is adequate for the problem or proves the conceptual model is correct, then no further study is needed. If not and funding is available, revise the conceptual model, collect more data and incorporate the data into a more complex model, recalibrate, and test the new hypothesis (from Kuniansky and others, 2022).

2 Installing MODFLOW

This text provides several hands-on exercises using MODFLOW. To do the exercises, you will need to download and install MODFLOW and other software by clicking on the link in each of the following bullets. Each software code was developed by the United States Geological Survey (USGS) and the links each connect to the web page for the software where the reader can select the appropriate download for their computing environment.

- MODFLOW↗, a groundwater flow modeling software.
- ModelMuse↗, one of the several graphical user interfaces for MODFLOW and the one for which step by step procedures are provided herein. (If desired, the exercises can be completed using a different graphical user interface.)
- MODPATH↗, a post-processing code that uses the output of MODFLOW to track flow lines. It is also available by clicking here↗.
- ListingAnalyst↗ (optional), a text file viewer that provides a table of contents and index for the MODFLOW listing file. ListingAnalyst is not required for the exercises; the MODFLOW listing file can be viewed with any text editor.

Compiled versions of MODFLOW for the Linux and MacIntosh operating systems are available here↗ if needed. All of these tools are provided by the USGS and all of them can be found on MODFLOW and Related Programs↗ page, except for ListingAnalyst.

2.1 Installing MODFLOW and MODPATH

MODFLOW 6 does not come with an installer. Instead, unzip the distribution file, mf*.zip, where * is the version number, and retain the directory structure in the distribution file. The preferred location for the unzipped directory structure is C:\WRDAPP (WRDAPP is an abbreviation for Water Resources Division Applications). Assuming you use the preferred installation location, you will have a directory like the one shown in Figure 2. The uppermost directory (mf6.3.0 in Figure 2) name will differ if you have downloaded a different version of MODFLOW 6. The MODFLOW executable itself (named mf6.exe) is in the "bin" subdirectory. For more about what is included in the distribution folder, see Box 2↴. MODPATH can be installed in the same way as MODFLOW.

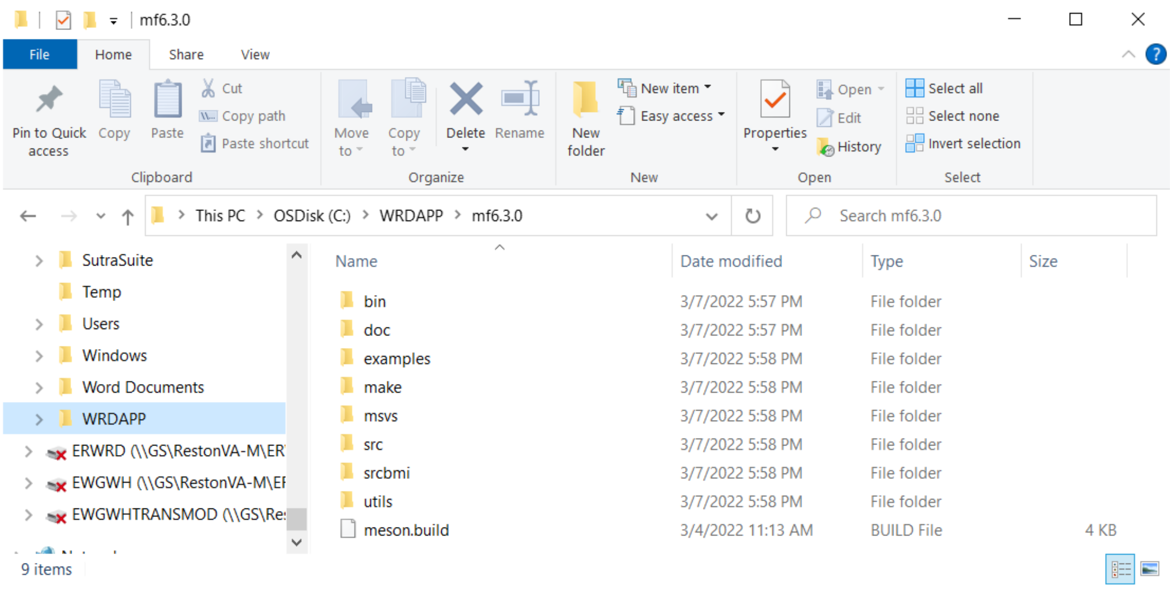


Figure 2 - The MODFLOW 6 distribution folder.

2.2 Installing ModelMuse

The exercises in this book use ModelMuse (Winston, 2019) to create the MODFLOW input files. To install ModelMuse, double click on the downloaded ModelMuseSetup*.exe file, where * represents the 32 or 64 bit operating system followed by the version number. Typically, the installer installs ModelMuse without complications.

There are several other graphical user interfaces for MODFLOW. You can find them on the Wikipedia page for MODFLOW↗. Besides using a graphical user interface, you can use Python to create and run MODFLOW models using Flopy↗.

2.3 Installing ListingAnalyst

ListingAnalyst is a text file viewer for MODFLOW listing files. You can download it here↗. It can be installed by unzipping its zip file into an empty directory. Its use will be described in the exercises of this book.

3 Orientation to ModelMuse

ModelMuse is a graphical user interface for MODFLOW and several other model codes. It has some functionality of a Geographic Information System (GIS). With GIS functionality, geometric objects and/or formulas can be created based on a global geographic coordinate system and then converted to model coordinates for generation of model input files. ModelMuse will be used for the exercises in this book. This section provides a quick orientation for readers to become familiar with ModelMuse. Later sections provide step-by-step instructions where readers can open and use the software while reading this text. However, it does not cover every ModelMuse feature. More information is available in the help system for ModelMuse. Links to online instructional videos about using ModelMuse are provided on the ModelMuse web page (https://www.usgs.gov/mission-areas/water-resources/modelmuse-tutorial-videos-0↗).

3.1 Defining the Initial Grid

When creating a new MODFLOW model in ModelMuse, after some initial screens, the Initial Grid screen appears where the grid and the preferred version of MODFLOW can be specified (Figure 3). There are several other ways to define the grid in ModelMuse but this is the most basic approach. After finishing creation of the grid, the main ModelMuse window appears.

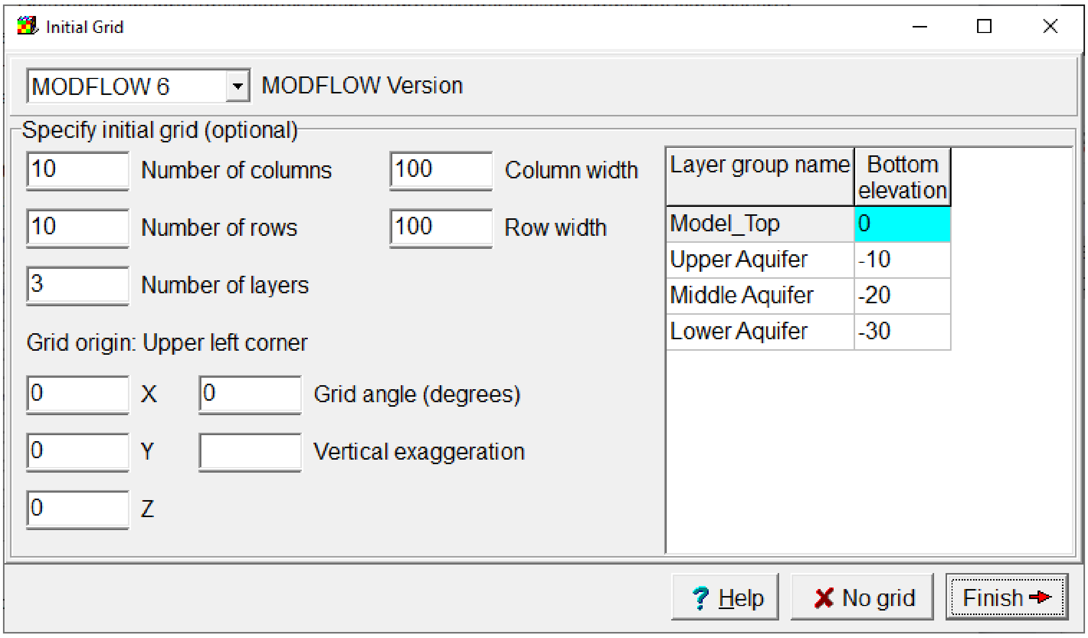


Figure 3 - Defining the initial grid in ModelMuse.

3.2 The Main ModelMuse Window

The main ModelMuse window presents various views of the model grid as shown in Figure 4, in which some of the most important buttons are labeled. The line of text across the top is a series of drop-down menus; *File*, *Edit*, *Grid*, *Data*, *Object*, *Navigation*, *View*, *Customize*, *Model*, *Model Selection*, and *Help*. The notation used in instructions for selecting an option from a drop-down menu is *Menu Name|Selection*, for example *Model|MODFLOW Packages and Programs*.

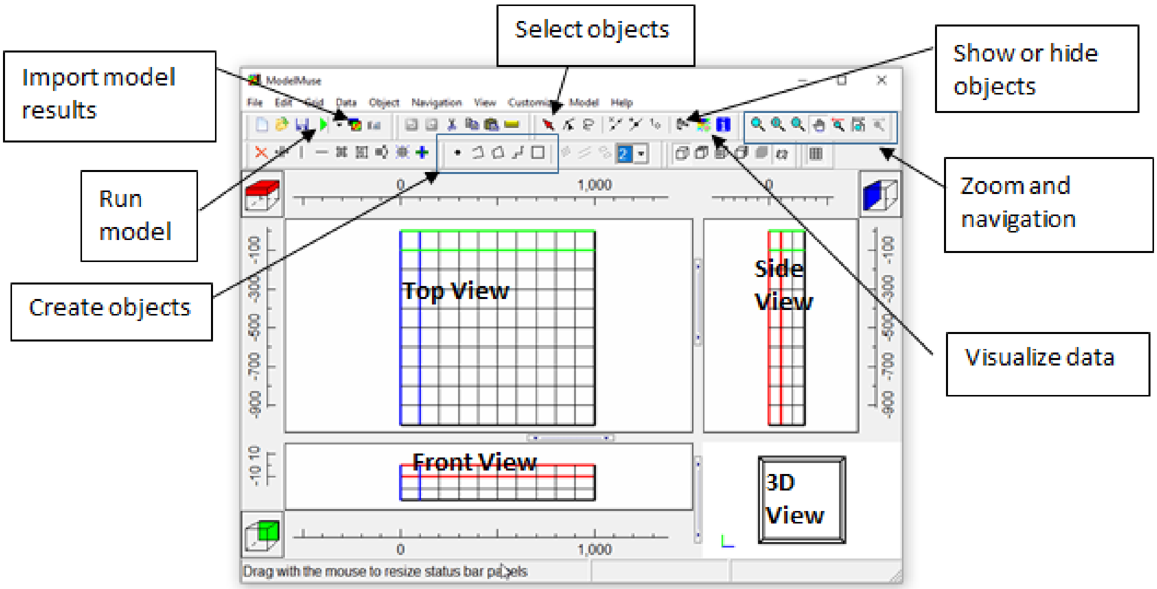


Figure 4 - The ModelMuse Main Screen.

3.3 Activating MODFLOW Packages

MODFLOW has a variety of packages that can be used to simulate different features in a model such as rivers or recharge. Packages are activated or deactivated by selecting *Model|MODFLOW Packages and Programs* and checking or unchecking the boxes for each package (Figure 5).

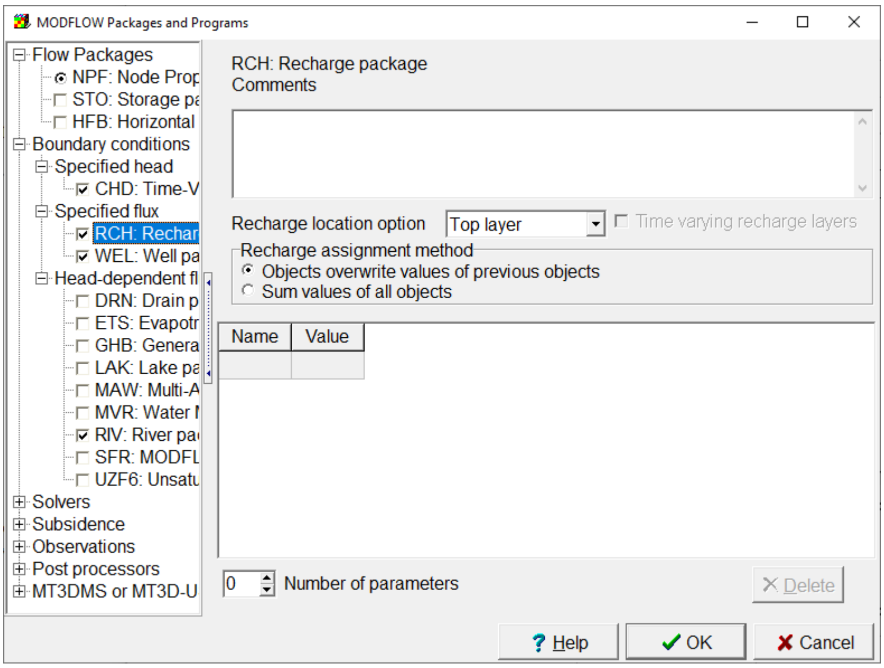


Figure 5 - The MODFLOW Packages and Programs Dialog Box.

3.4 Data Sets

Depending on which packages are selected, different data sets will be created. The values assigned to these data sets can be specified in the *Data|Edit Data Sets* dialog box (Figure 6). Additional data sets can be created that are specific to the way you want to build your model.

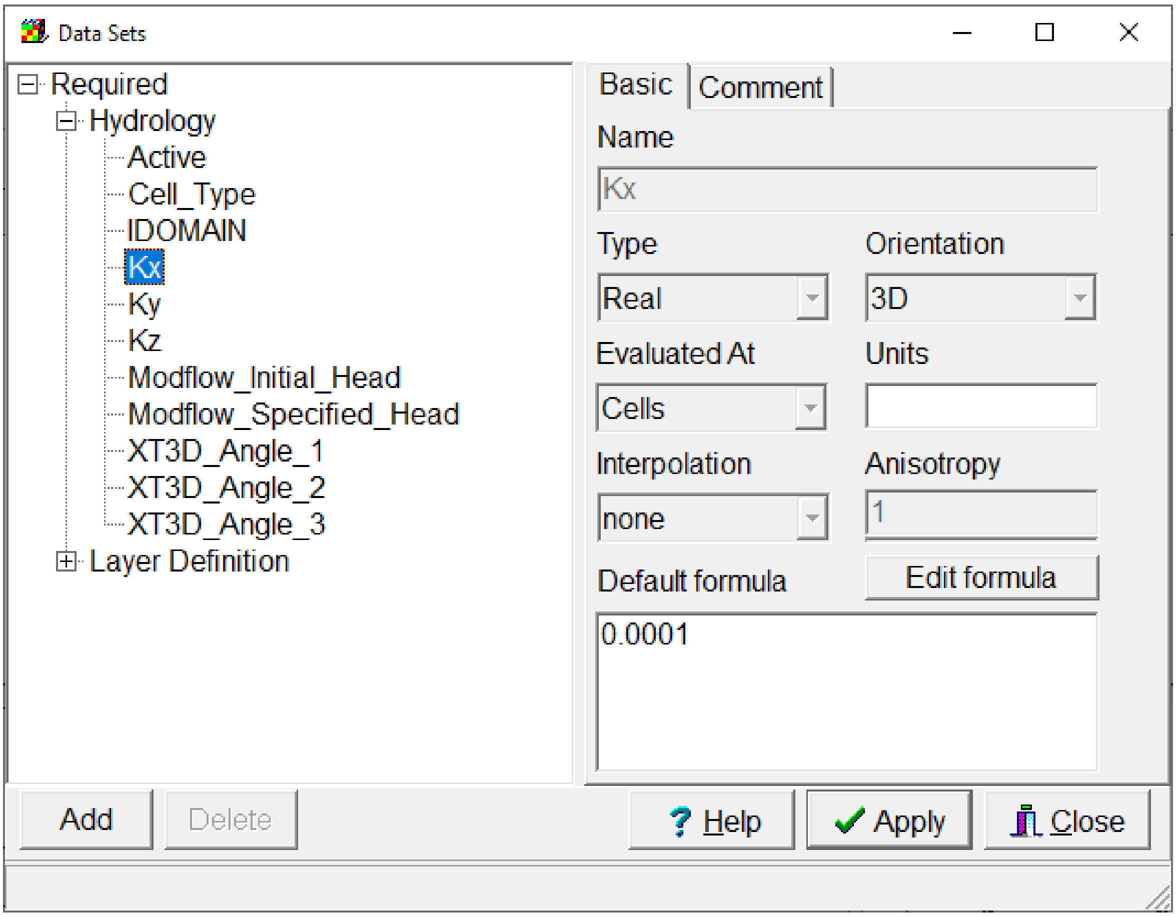


Figure 6 - Data Sets Dialog Box.

Numeric values can be assigned to data sets. Alternatively, formulas can be input that result in a distribution of different values throughout the model grid-cell locations. For example, it is often convenient to assign the starting head of a MODFLOW model to be identical to the top of the model as that can help ensure that none of the cells will go dry (i.e., have a head below the bottom of the cell) on the first iteration of the first time-step. The Formula Editor (Figure 7) is used to help create properly formatted formulas.

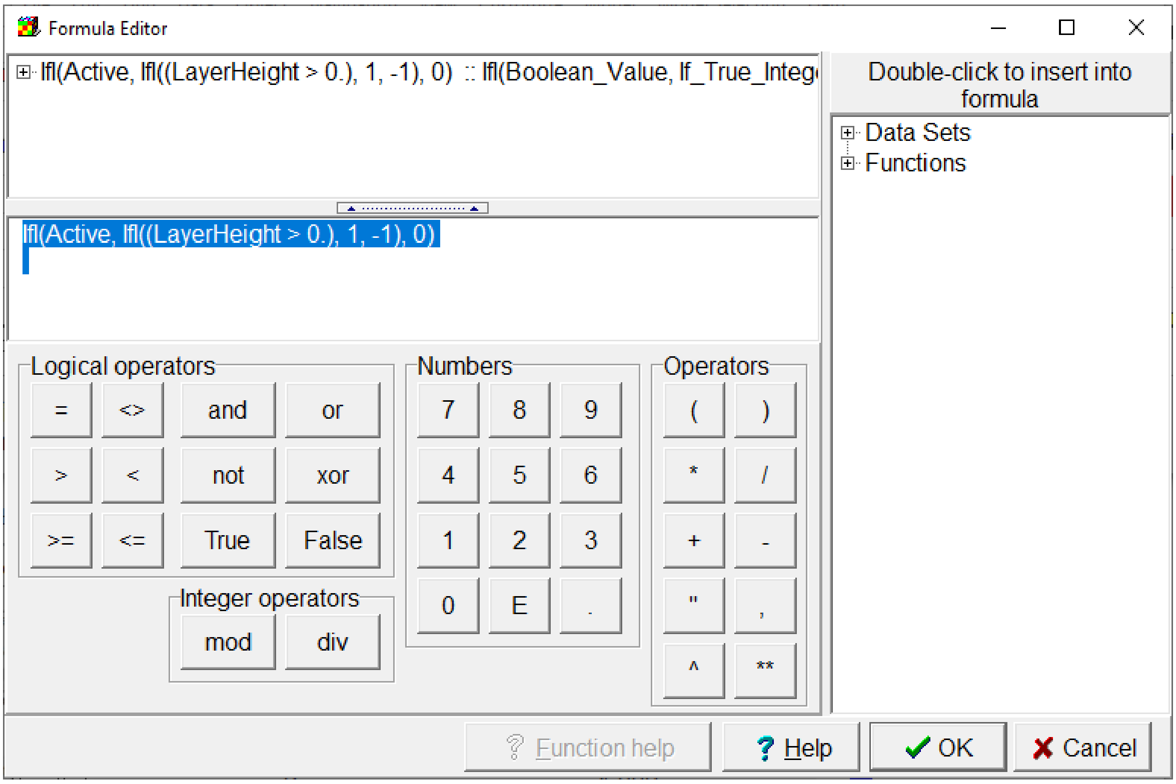


Figure 7 - Formula Editor showing the default formula for the IDOMAIN data set. The IDOMAIN data set in MODFLOW is used to distinguish between active cells (IDOMAIN >0), inactive cells (IDOMAIN = 0), and vertical pass-through cells (IDOMAIN < 0). In this example, IDOMAIN depends on the “Active” data set and the height of the cell. Inactive cells are assigned a value of 0. Active cells that have layer heights less than or equal to zero are assigned a value of -1. Other active cells are assigned a value of 1.

3.5 Data Visualization

The *Data|Data Visualization* dialog box (Figure 8) is used to color the grid with the values of a data set or model feature or to specify contour lines for a data set.

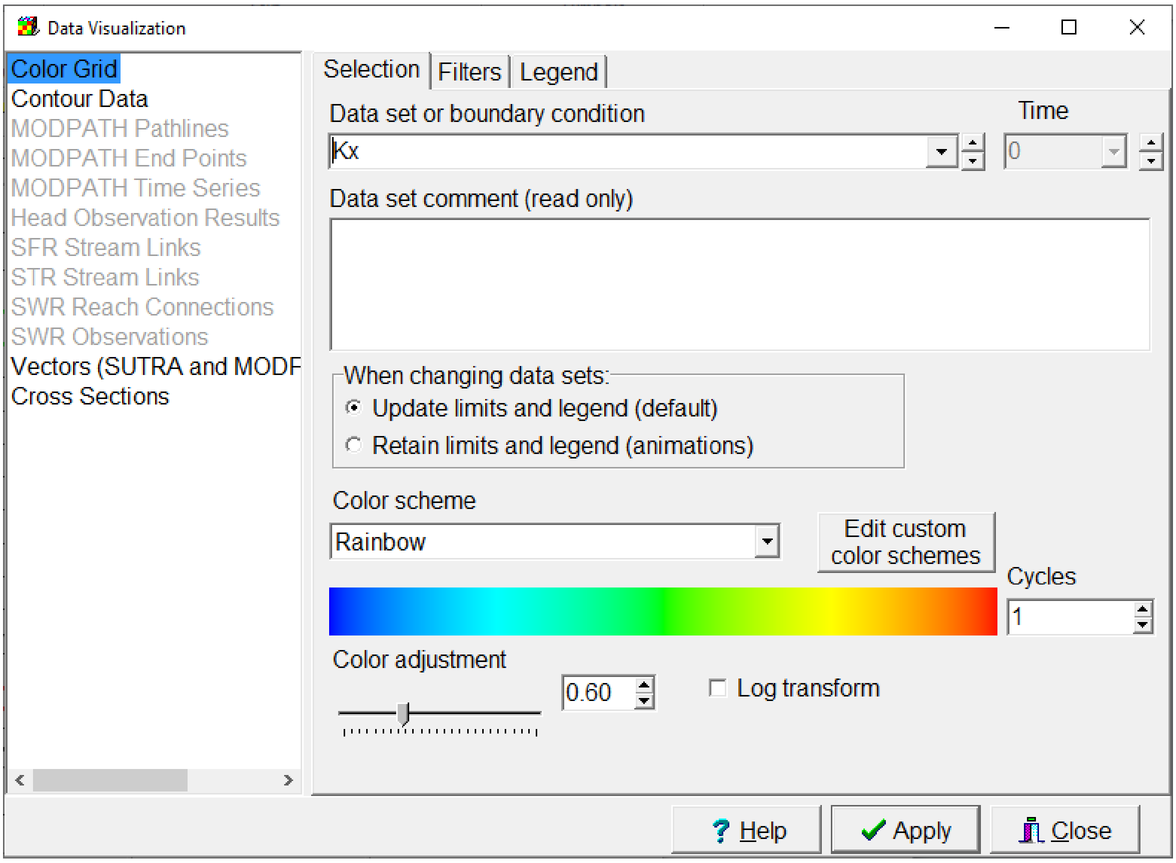


Figure 8 - Data Visualization dialog box.

If you are unsure about how a data set value has been assigned, the *Data|Show Grid or Mesh Values* dialog box can help (Figure 9). When the cursor is placed over a cell, the dialog box shows not only the value of the data set in that cell but an explanation of how that value was assigned.

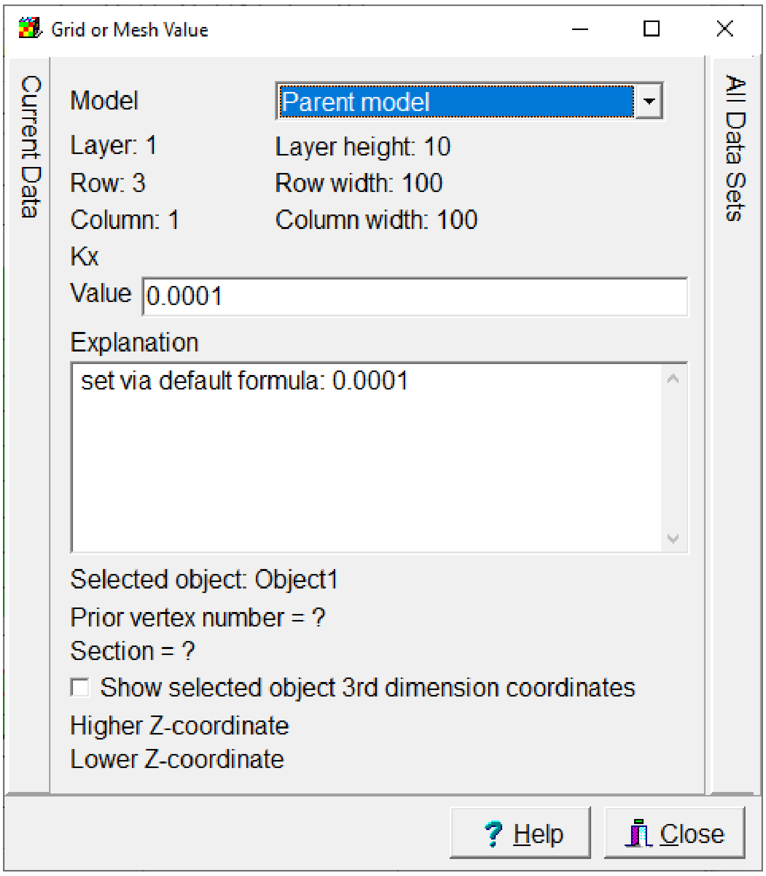


Figure 9 - Grid or Mesh Values dialog box. A Kx value is shown because Kx has been visualized using the Data Visualization dialog box and the cursor has been placed at layer 1, row 3, column 1.

3.6 Run MODFLOW

After creating all the required data sets and including some boundary conditions, the modeler can run the model by selecting *File|Export|MODFLOW 6 input files*. ModelMuse typically runs MODFLOW from within the ModelMonitor program (Figure 10). ModelMonitor displays the percent discrepancy in the water budget for each time step of the model simulation. The water budget accounts for all the water entering and leaving the groundwater system and for changes in the amount of stored water. Ideally, these would all balance, but because of the iterative way that MODFLOW calculates the flows, there is typically a small discrepancy in the budget.

While ModelMonitor is running, it checks for error or warning messages. If everything is going well, a green smiling face is displayed on the left end of the ModelMonitor form. If the program detects any warning messages, a yellow neutral face is displayed. If it detects an error message or the percent discrepancy is too high, a red frowning face will be displayed. This can help the modeler identify problems with the model.

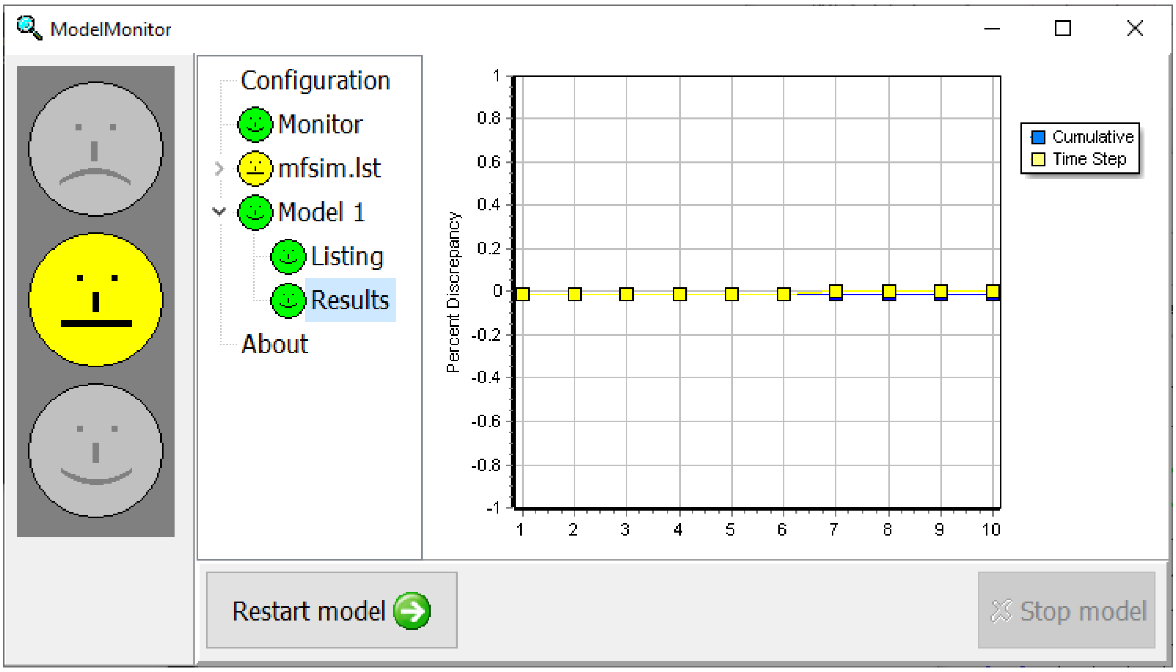


Figure 10 - ModelMonitor program running MODFLOW.

3.7 Objects

Objects are the primary means for specifying spatially variable data in ModelMuse. Objects are created by selecting one of the buttons for creating objects (Figure 11), then clicking on the top, front, or side view of the model. For line or polygon objects, each click adds another vertex and double-clicking finishes the object.

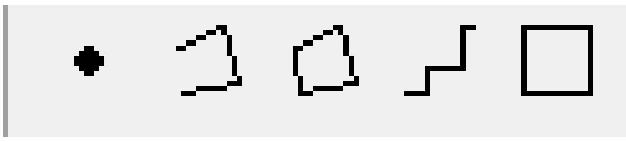


Figure 11 - Buttons for Creating Objects. From left to right an object can be a point, polyline, polygon, straight line, or a rectangle.

When the shape of the object is complete, the Object Properties dialog box appears (Figure 12a) in which values can be specified for data sets or model features ((Figure 12b) such as rivers or wells.

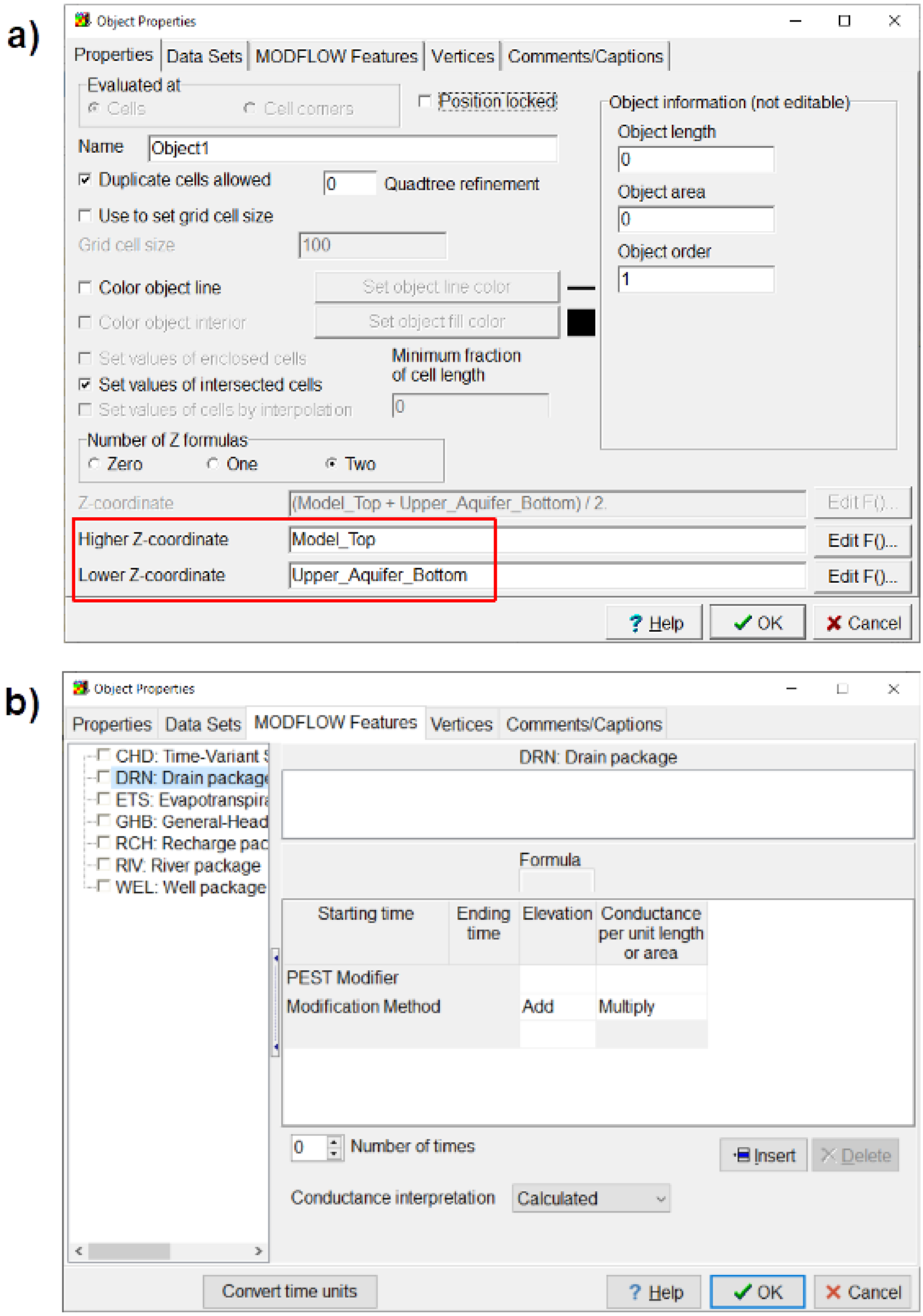


Figure 12 - Object Properties dialog box. a) Properties tab with Z formulas highlighted. b) MODFLOW Features tab. If you are not familiar with MODFLOW and MODFLOW packages or parameter estimation programs like PEST, this dialog box may be confusing. Its use is demonstrated in several of the exercises in this book. For the object properties shown here, you may want to change the Name of Object1 to reflect to what the object does. Since the Drain package (DRN) is highlighted, this object could be a linear feature for an ephemeral stream or a point feature for a spring that can go dry.

For objects drawn on the top view of the model, the formulas for the Z-coordinates (Figure 12a) determine which layer or layers the object affects.

Objects can be hidden or displayed using the *Object|Show or Hide Objects* dialog box.

To edit an object after it has first been created, select *Object|Select Objects* and then double-click on the object.

3.8 Import Model Results

To visualize the model results, select *File|Import|Model Results*, then select the output file for the model. This brings up the dialog box shown in Figure 13. Any or all the data sets (e.g., some, all, or one timestep) can be selected for import. You can also choose to use one of the data sets to color the grid or to create contour lines.

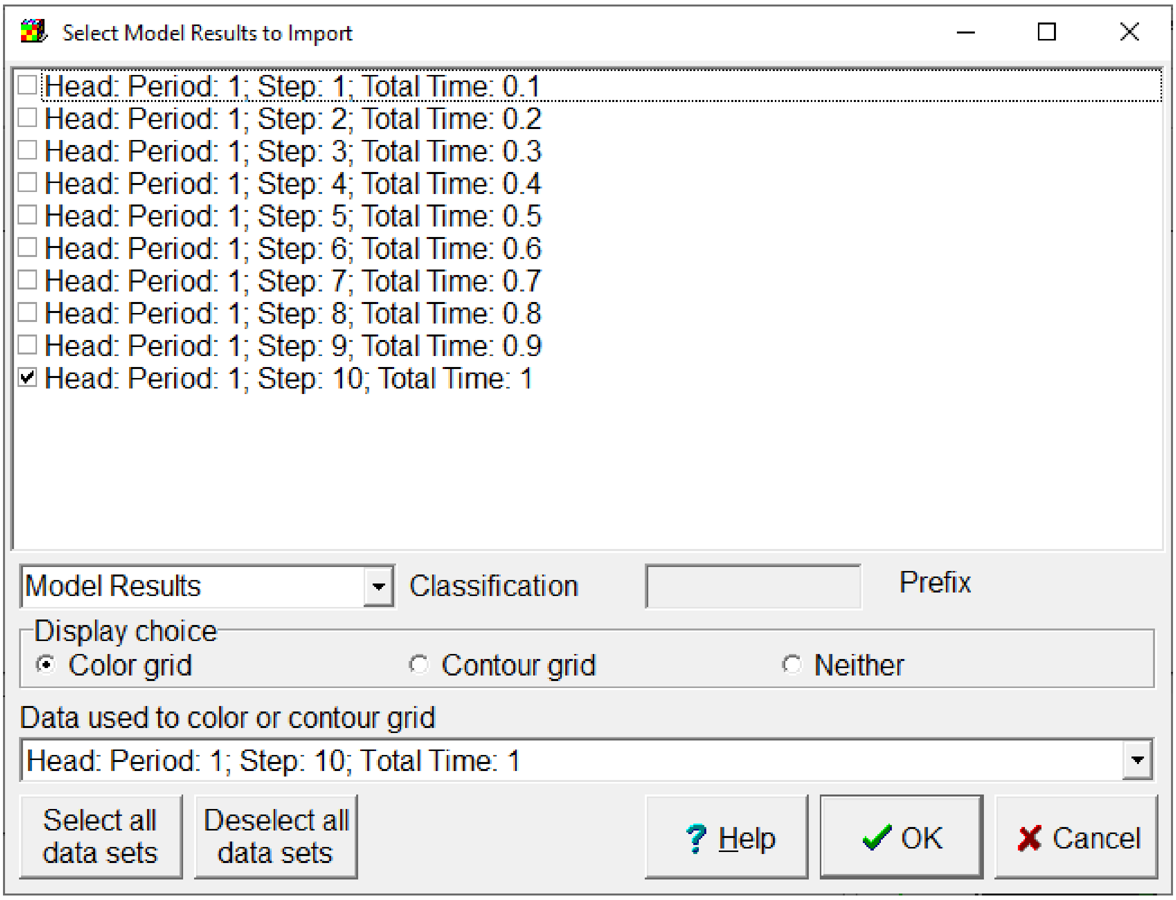


Figure 13 - Importing model results.

4 Creating a Simple MODFLOW Model

It is possible to create the input files for MODFLOW using only a text editor. However, generating the model input files with only a text editor requires significant effort to understand the formats of the various files and to generate them correctly. This book relies on ModelMuse to generate the input files. More information about the formats of MODFLOW input files is provided in documents in the doc subdirectory of the MODFLOW software folder, described in Box 2. For older versions of MODFLOW, information about input formats is provided by the Online Guide to MODFLOW↗.

4.1 Run an Example MODFLOW Model

Before creating a MODFLOW model in ModelMuse, try running one of the example models from the MODFLOW 6 distribution file. Open the Examples folder from the distribution and then go to one of its subfolders. Each subfolder contains one example model. Locate the file in the subfolder named “run.bat” and double-click it to start running the model. By default, Windows does not show filename extensions, so the file name may appear to be “run” instead of “run.bat.” Depending on the security settings, the computer may open a window that notes the application is not recognized and wait for you to click that it is safe to proceed. When the run file executes, a command line window will open and MODFLOW will run (Figure 14).

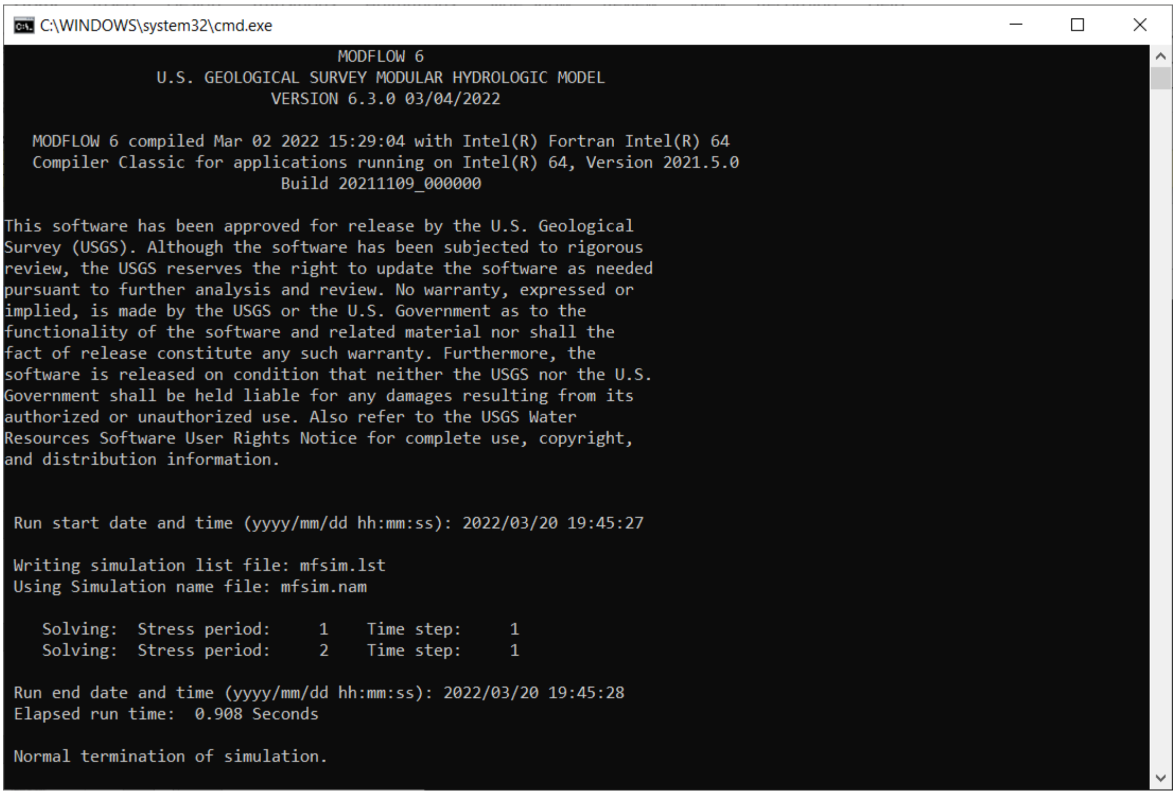


Figure 14 - Screen capture of MODFLOW running in a command-line window.

This is not a particularly intuitive way of running a computer program. In this book, MODFLOW will not be executed in this way. Instead, a graphical user interface will run MODFLOW. Although MODFLOW may look different when run through a graphical user interface, the MODFLOW software is separate from the graphical user interface. The ModelMuse interface executes MODFLOW by running a .bat file that runs ModelMonitor, and ModelMonitor runs MODFLOW as a background process. Other interfaces use other methods for running MODFLOW.

4.2 Model Requirements

MODFLOW requires that the modeler define nodes covering the area or volume to be modeled. The following example will use a rectangular grid with nodes representing the center of each cell. MODFLOW calculates the heads and flows at each node. The time period to be simulated also needs to be specified. The model simulation time is broken down into "stress periods" that are further divided into time steps as described in Section 5.3. The stress periods define time periods within which stresses on the system, such as pumping from wells, recharge, or river stage, remain constant (unless Time-Variable Input is used as described in Section 10). The modeler must also specify the stresses on the system and the relevant properties of the rock or sediments through which the flow occurs. Additionally, specified head or head-dependent boundary conditions should be defined at locations within the model domain based on known elevations of, for example, drains or rivers. If heads are specified everywhere, the solution for head would be completely constrained and there would be no point in running MODFLOW. If no specified heads, head-dependent boundaries, or fluxes are specified, the MODFLOW model will be under-constrained and a solution cannot be obtained.

4.3 Starting a New MODFLOW Model

To begin, we will create a very simple model. It will have 1 layer, 30 rows, and 30 columns. There will be two specified head boundaries in opposite corners.

- Start ModelMuse by double-clicking on its shortcut in the computer's Start menu.
- Select *Create new MODFLOW model* and click *Next* (Figure 15).

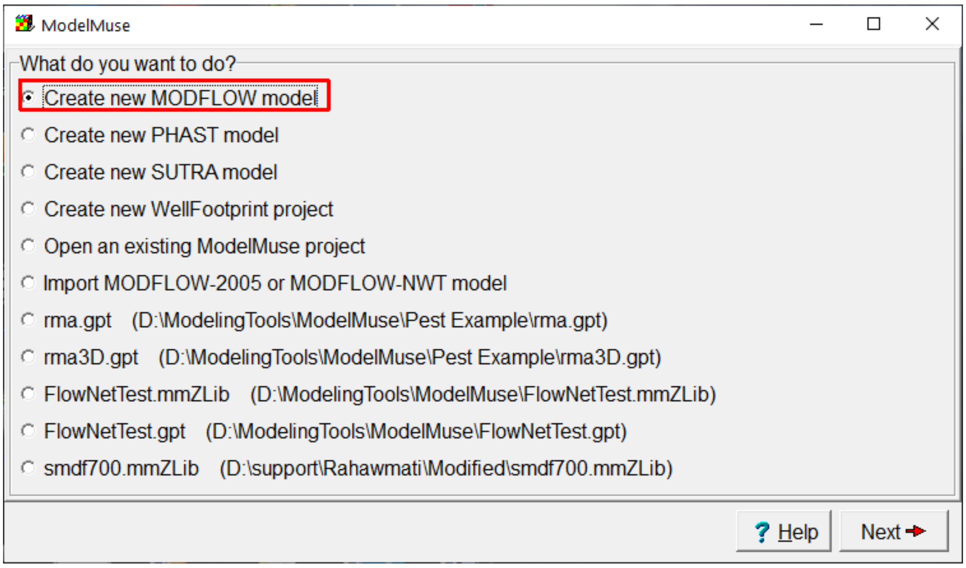


Figure 15 - Selecting *Create new MODFLOW model* to start a new MODFLOW model in ModelMuse.

- (Optional) Enter a description of the model. (Figure 16).
- Leave the length and time units set to meters and seconds.
- Set the projection to "N/A" (not applicable) because this is a generic model example. For a model of a specific groundwater system, enter the geographic projection used to associate the model coordinates with its location on Earth.
- Click *Next*.

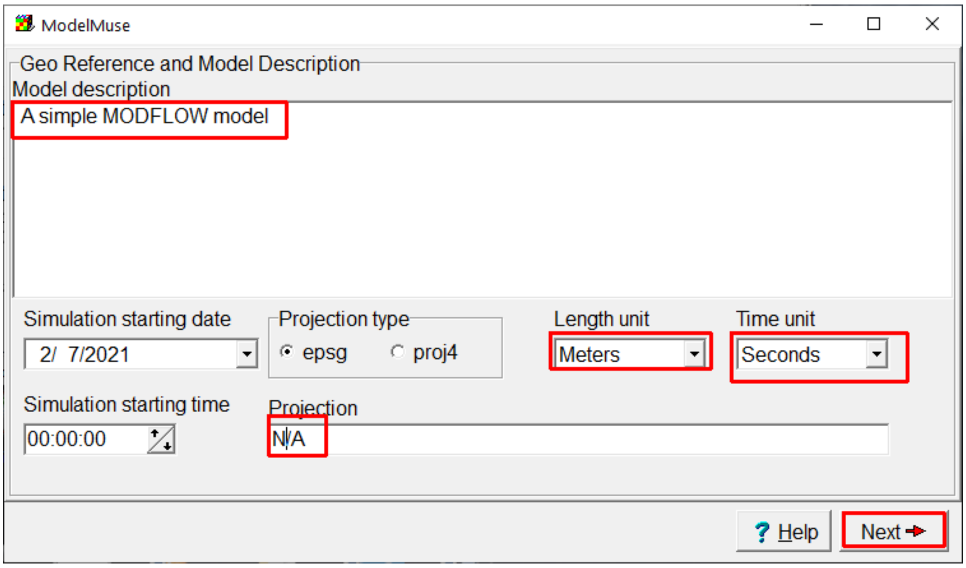


Figure 16 - Specifying model description and coordinate measuring system for a new MODFLOW model in ModelMuse. Enter the model description and since no geographic coordinate is used, Projection should be entered as "N/A" for not applicable. Then click on *Next* to go to the next screen to edit information about the model.

- Set the MODFLOW version to MODFLOW 6 (Figure 17).
- Set the number of rows and columns to 30 and the number of layers to 1. Set the column and row spacing to 1000. Set the top of the model to 100 and the bottom of the aquifer to -50. Then click *Finish*.

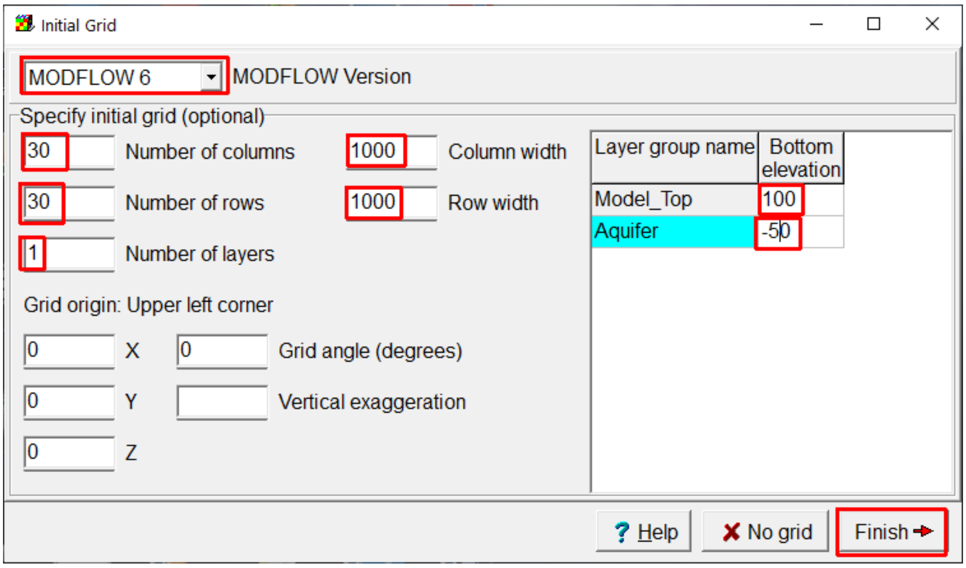


Figure 17 - ModelMuse Initial Grid dialog box showing one way to define a new MODFLOW grid. Edit all the red highlighted cells as shown if needed. Then click on *Finish*.

The main ModelMuse form will be displayed with top, front, side, and three-dimensional (3D) view of the model (Figure 18).

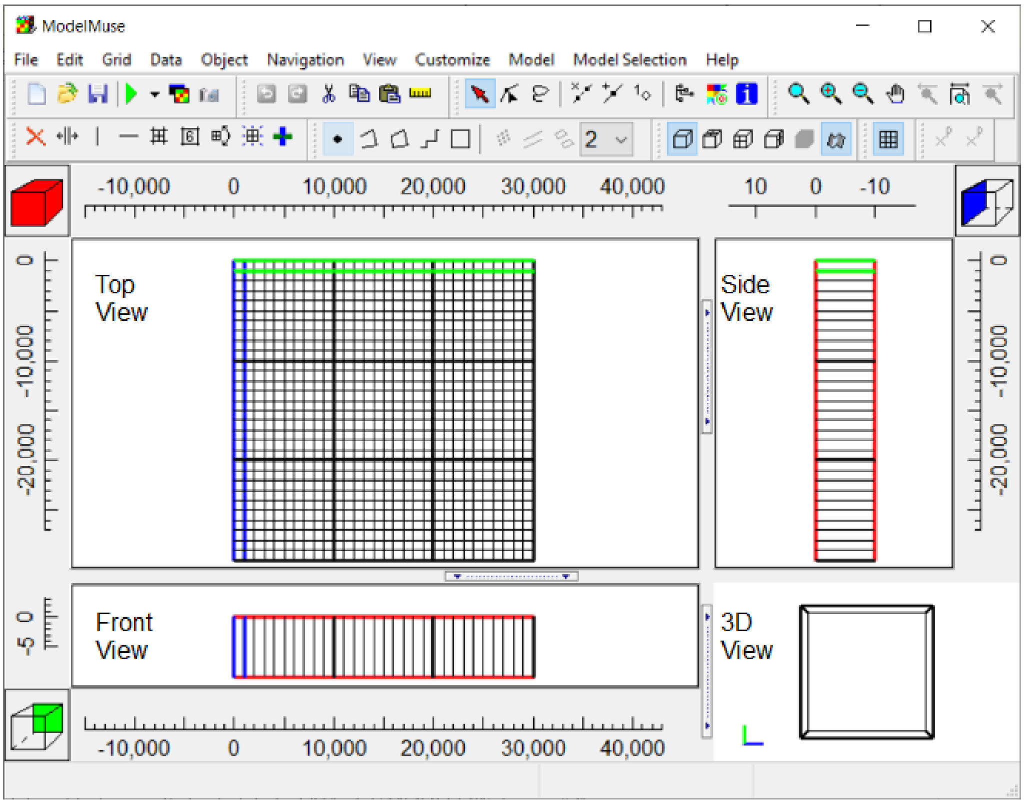


Figure 18 - Model grid as displayed by ModelMuse in the top, front, side, and 3D views of the model.

4.4 Define Stress Periods

The next step is to define the stress period. In this case, the stress period is already defined but we will check it. Along the top of the ModelMuse window are drop down menus: *File*, *Edit*, *Grid*, *Data*, *Object*, *Navigation*, *View*, *Customize*, *Model*, *Model Selection*, and *Help*. Click on *Model* and select *MODFLOW Time…* from the drop-down menu.

One steady-state stress period is defined with one time-step. Click *Cancel* to close the dialog box (Figure 19).

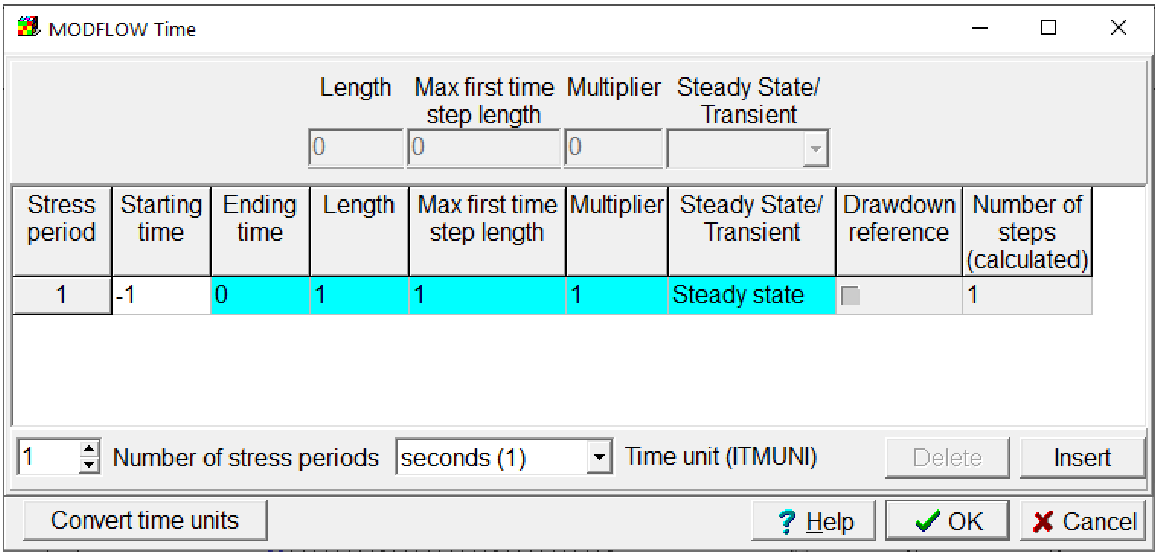


Figure 19 - ModelMuse MODFLOW Time dialog box showing the definition of a single stress period.

4.5 Specify Hydraulic Conductivity

The next step is to set the value of hydraulic conductivity. Select *Data|Edit Data Sets….* Then expand *Required* by clicking on the plus sign. Next expand *Hydrology* and select *Kx* which represents the hydraulic conductivity in the X direction. Check that the Default formula is 0.0001 (Figure 20). You can also check the Active data set. It specifies the cells that are active. At present, it should be set to “True,” which means that every cell is active. Next check *Ky* and see that the default formula for Ky is “Kx” meaning that, by default, the system is isotropic in the x,y direction of the model layer. Now if you check *Kz*, it will show that the default assumes vertical hydraulic conductivity using a formula for Kz of “Kx/10.” Click on *Close* to close the dialog box.

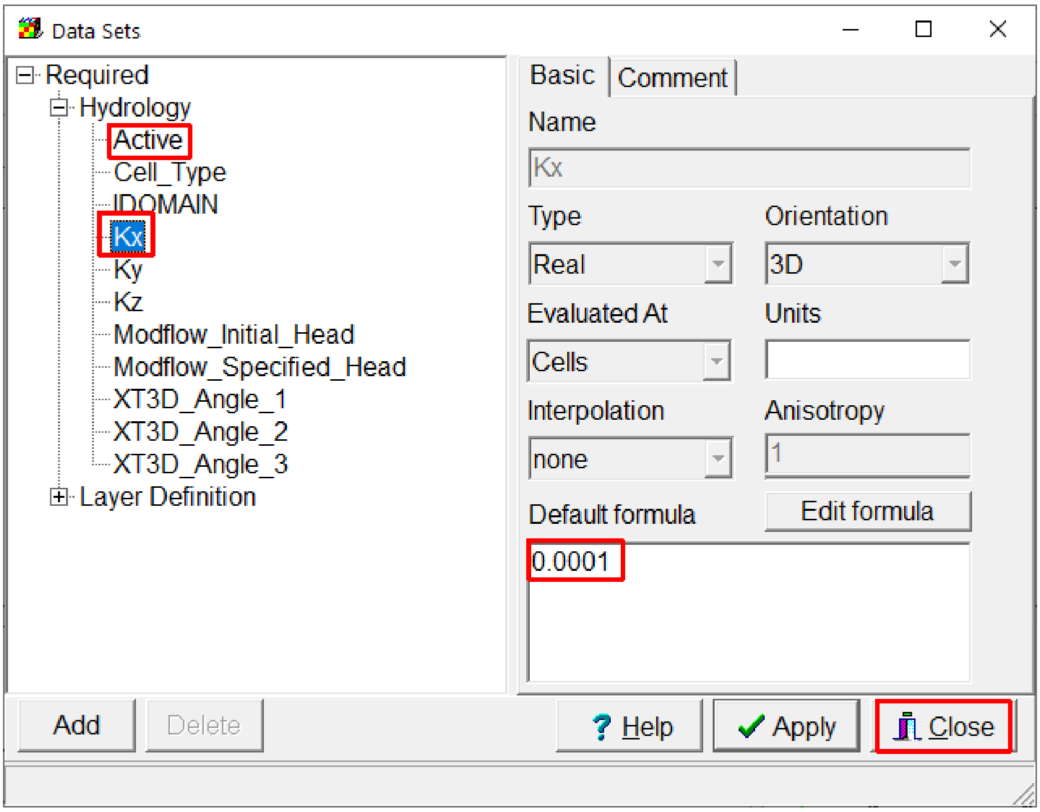


Figure 20 - ModelMuse Data Sets dialog box showing default formula for the hydraulic conductivity in the X direction (Kx).

4.6 Defining Specified Head Boundaries

Each type of boundary in MODFLOW has a corresponding "package." To use a boundary of a particular type, activate the corresponding package. This model will have specified head boundaries. Those are defined with the Constant Head (CHD) package. In earlier versions of MODFLOW, this was called the "Time Variant Specified Head Package" which is a better reflection of how the package operates.

To activate the CHD package, select *Model|MODFLOW Packages and Programs…*. Expand *Boundary Conditions*, expand *Specified Head* and check the *CHD* box. Click *OK* to close the dialog box (Figure 21).

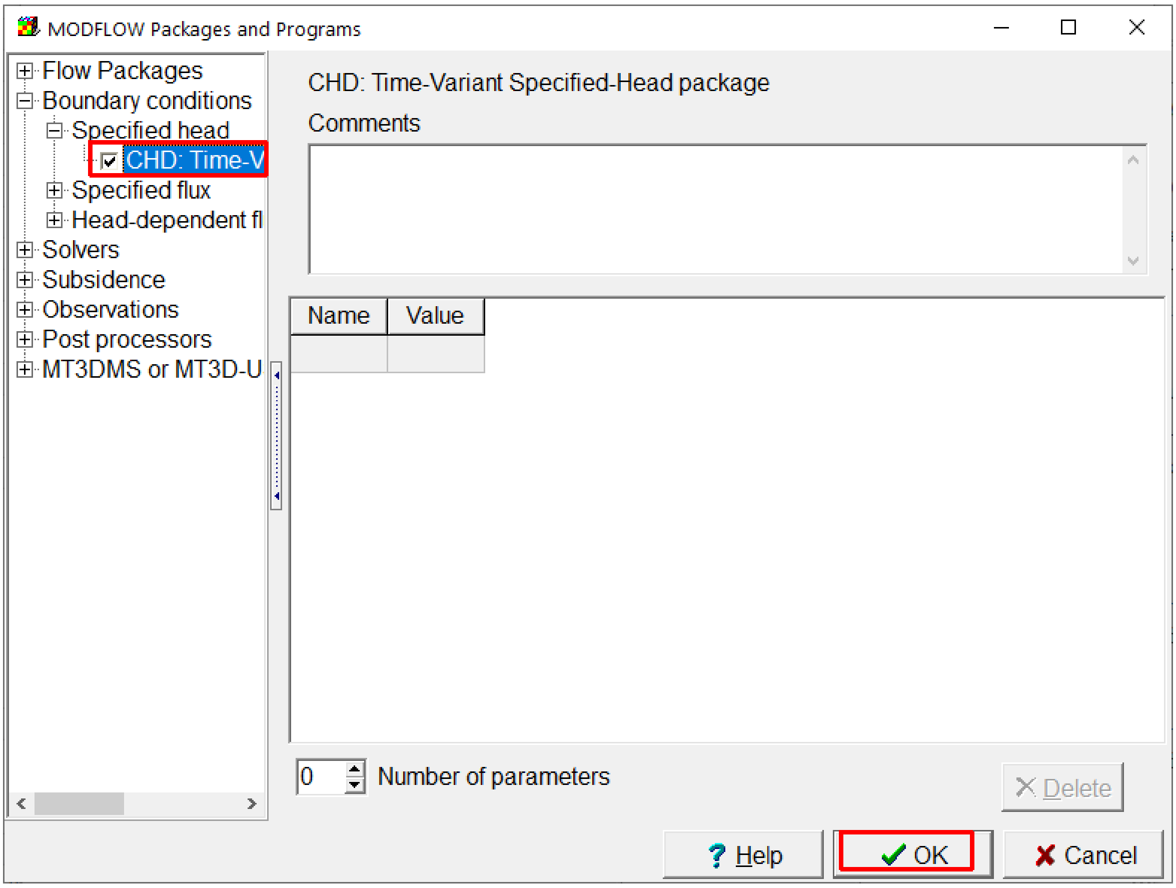


Figure 21 - Activating the CHD package in the "MODFLOW Packages and Programs" dialog box in ModelMuse.

Next, create the specified head boundary. Click on the *Create point object* button and then click in the cell in the upper left corner of the grid. The Object Properties dialog box will appear (Figure 22). Go to the MODFLOW Features tab and check the checkbox for the CHD package, then set the Number of times to 1. In the 3rd line of the table, enter a starting time of -1, and an ending time of 0. Set the starting head to 110. The starting head will be the head from time -1 to time 0. The ending head is not used in MODFLOW 6 but was used in earlier versions of MODFLOW. Click *OK* to close the dialog box. (By default, ModelMuse sets the starting and ending times for the initial steady-state stress period to -1 and 0 respectively. If subsequent transient stress periods are added, this results in the transient stresses starting at time = 0.)

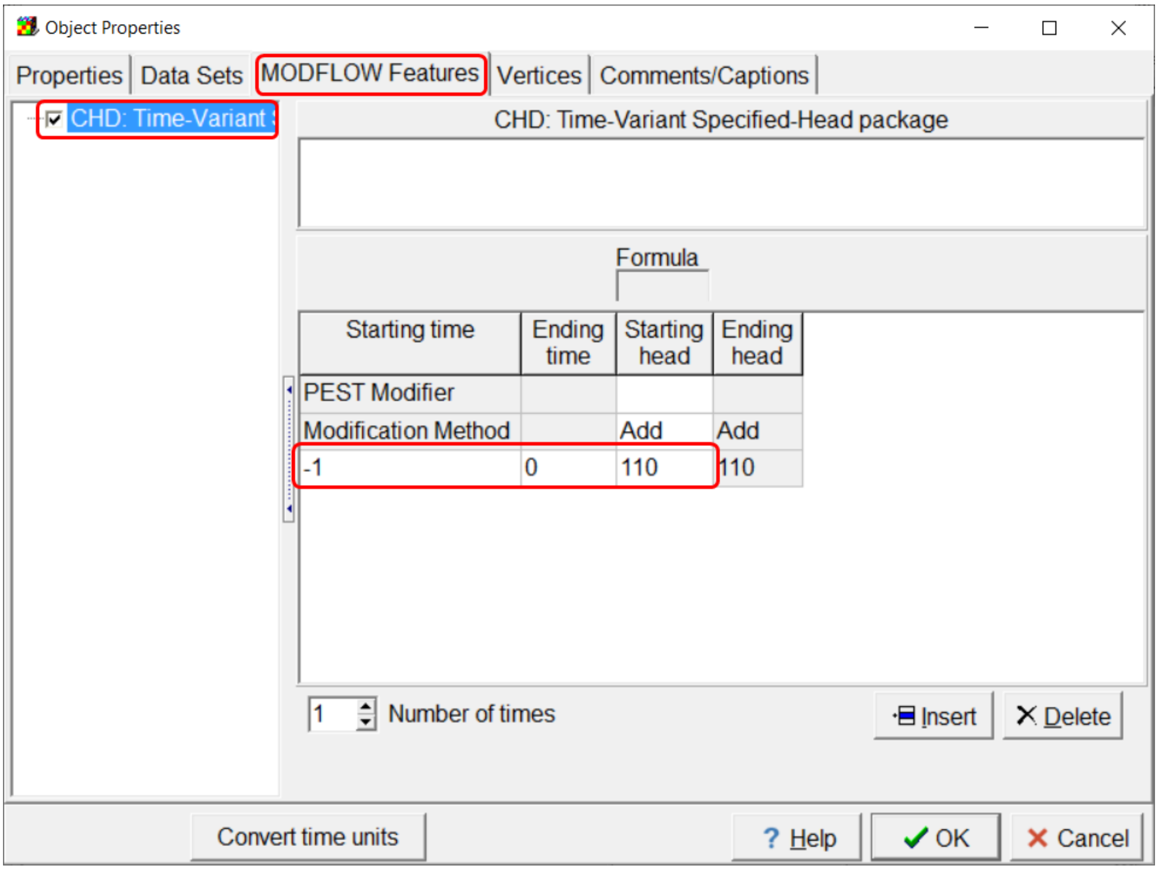


Figure 22 - Specifying a specified-head boundary in the Object Properties dialog box.

Create another point object in the lower right corner of the grid and use it to specify another CHD boundary. This time, however, set the starting head to 100.

It would be a good idea to save your work now. Select *File* | *Save* and save the model to an empty directory on the computer with a directory and file name of your choice. There are two checkboxes on the Save dialog box labeled "Create archive" and "Save data set values." If "Create archive" is checked, ModelMuse will create a zip file containing the model input and output files as well as the ModelMuse files and other related files. If "Save data set values" is checked, The ModelMuse file will contain the values of any up-to-date data sets so that they do not have to be recalculated when opening the ModelMuse file.

4.7 Running the Model

Now click on the *Run* button. A *Save* dialog box will appear asking you to save a "Name file." This is a main input file for MODFLOW that organizes most of the other files needed to run the flow model. You can save it in the same directory in which you saved your model or in a different directory. (There is also another name file that organizes the entire simulation. It is always named mfsim.nam.)

If ModelMuse doesn't know where you have MODFLOW installed, it will prompt you to specify its location on your computer. Otherwise, it will start running MODFLOW.

Two new windows will appear. The first will be a command-line window

(Figure 23). The second will be a ModelMonitor window (Figure 24). Both are separate programs from ModelMuse. ModelMonitor is started from the command-line window. ModelMonitor runs MODFLOW in the background and the ModelMonitor window displays some of its results on the screen. The contents of the ModelMonitor window are discussed in later sections of this book. When the *Stop model* button in the ModelMonitor becomes disabled (i.e., becomes grayed-out), MODFLOW has finished running and you can close the ModelMonitor window by clicking the x in the upper right corner. MODFLOW finishes very quickly for this simple model but can take a long time for complex models. The command line window will then start Notepad which will display the main output file (called the "listing file") from MODFLOW. The type of information presented in the listing file is discussed in later sections of this book. You can close the listing file when you are finished with it and then close the command-line window.

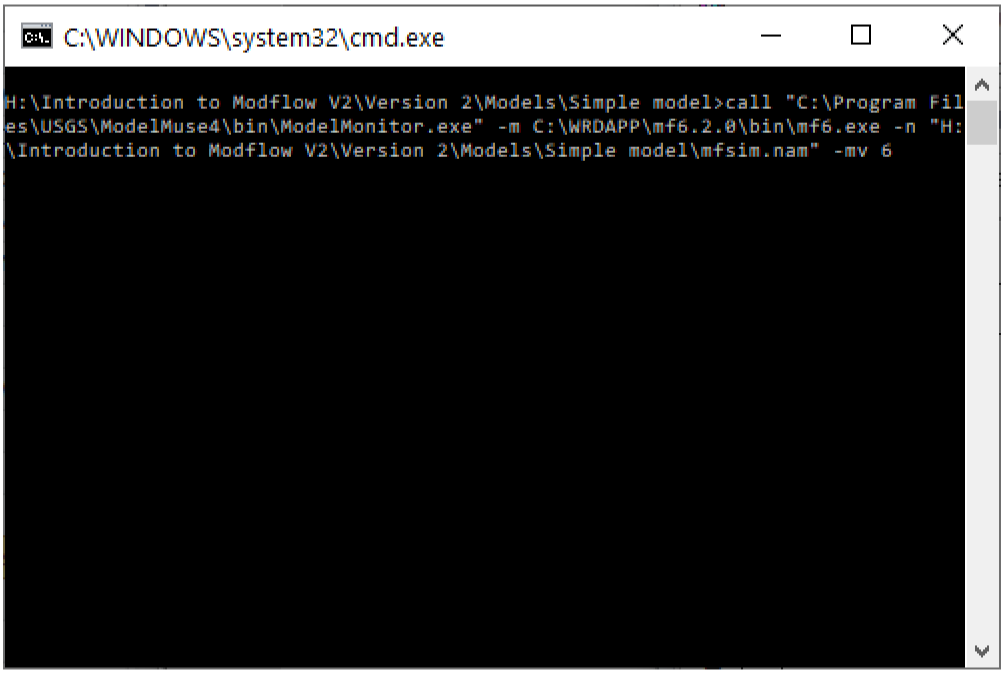


Figure 23 - Command-line window used for starting ModelMonitor.

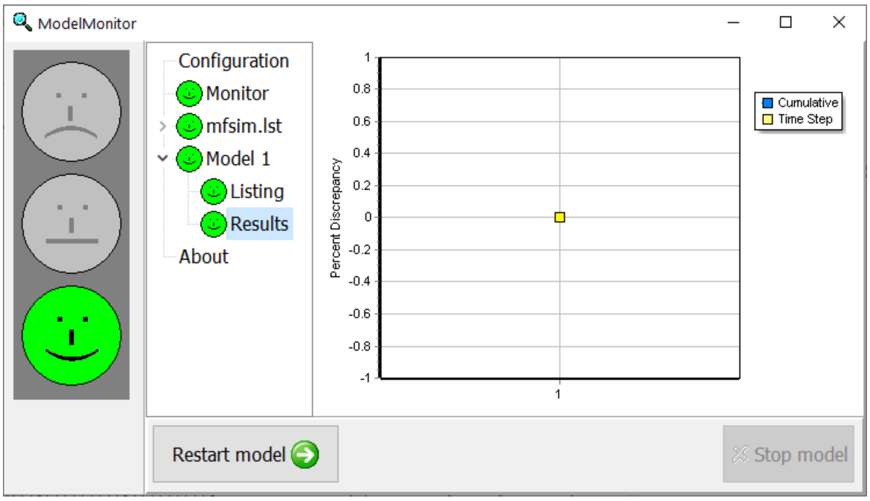


Figure 24 - ModelMonitor running MODFLOW.

4.8 View Model Results

Next, look at the results of the model. Select *File | Import | Model Results…* and a Select Model File dialog box will appear. One of the files created by MODFLOW will contain the calculated head at each cell. This file will have the extension ".bhd" which stands for "binary head." Select the file with the extension ".bhd" generated by your model and click the *Open* button.

Another dialog box will appear. It will list all the time steps for which data have been saved in the bhd file. This model only had one time-step so data for only one time-step have been saved in the file. Next, set the Display choice to *Color grid* and click on *OK*. The color coded, simulated distribution of hydraulic head should display as shown in Figure 25.

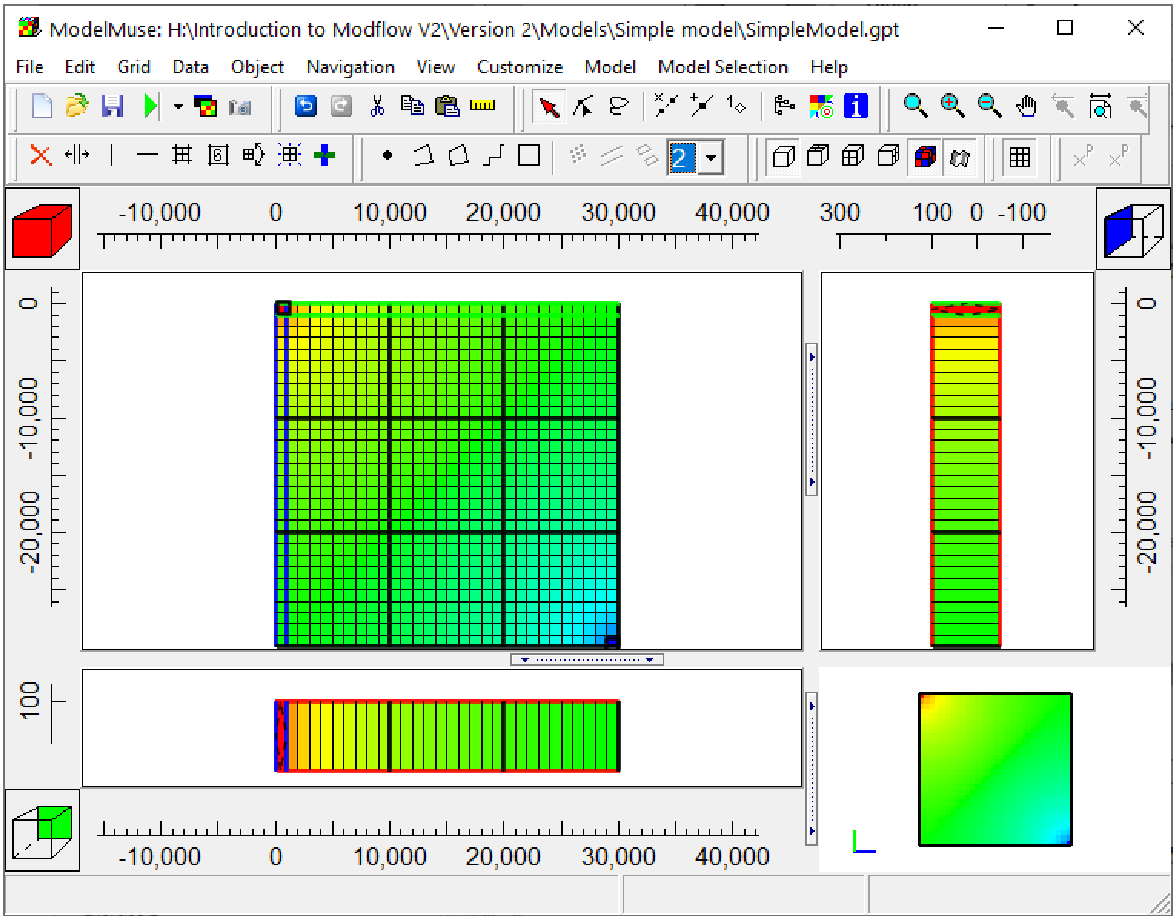


Figure 25 - ModelMuse main window with grid colored based on the simulated heads with red indicating a head of 110 m and blue indicating a head of 100 m.

4.9 MODFLOW Input and Output Files

In the previous section, a MODFLOW model was created, executed, and the model results were examined. Now let's look more closely at the input and output files for the model. The input files are all text files that you could create with a text editor. First find the folder where you saved the model, then open the RunModel.bat file in a text editor. It contains a single line of text (Figure 26). If you are running a different version of

MODFLOW 6, that version number will replace "6.3.0."

```
C:\WRDAPP\mf6.3.0\bin\mf6.exe
```

Figure 26 - Contents of RunModel.bat.

This line starts MODFLOW 6 (mf6.exe) running. The first thing MODFLOW 6 does is read a file named mfsim.nam. This is the "Simulation Name File." Open mfsim.nam in a text editor. We will not discuss the details of this file but notice that it contains the names of several other files used in the model (SimpleModel.tdis, SimpleModel.nam, and SimpleModel.ims) as shown in Figure 27.

```
BEGIN TIMING
  TDIS6 'SimpleModel.tdis'
END TIMING

BEGIN MODELS
  GWF6 'SimpleModel.nam' 'MODFLOW'
END MODELS

BEGIN EXCHANGES
END EXCHANGES

BEGIN SOLUTIONGROUP     1
  MXITER     1
  IMS6 'SimpleModel.ims' 'MODFLOW'
END SOLUTIONGROUP
```

Figure 27 - Contents of mfsim.nam.

Each file in mfsim.nam is preceded by an abbreviation indicating the function the file plays in the simulation. For example, "SimpleModel.tdis" is preceded by "TDIS6." In the MODFLOW 6 input instructions, TDIS6 identifies a file that provides the stress period and timestep information for the model. (TDIS stands for "time discretization.")

A simulation may contain more than one model. In this example, there is only one model in the simulation, and it has its own .nam file named "SimpleModel.nam." The contents of SimpleModel.nam are shown in Figure 28.

```
# Name File for MODFLOW created on 3/20/2022 by ModelMuse Version 5.0.0.0
BEGIN OPTIONS
  LIST SimpleModel.lst
  PRINT_INPUT
  SAVE_FLOWS
END OPTIONS

BEGIN PACKAGES
  DIS6         SimpleModel.dis  DIS
  IC6          SimpleModel.ic  IC
  NPF6         SimpleModel.npf  NPF
  OC6          SimpleModel.oc  OC
  CHD6         SimpleModel.chd  CHD-1
END PACKAGES
```

Figure 28 - Contents of SimpleModel.nam.

The main concept to recognize here is that a MODFLOW model consists of numerous files, not only one file. More information about the structure of these files is provided by mf6io.pdf which is located in the doc directory of the MODFLOW 6 distribution folder.

To learn more about the MODFLOW input files, the reader can explore Exercise 1↴.

5 Discretization in Time and Space

In MODFLOW, time and space are discontinuous. Space is broken up into blocks called cells. Heads are calculated only at the center of each cell. Flows are calculated at the interfaces between adjacent cells. Flows are also calculated between cells and boundary conditions. Boundary conditions are where water enters and leaves the model. Streams and wells are examples of boundary conditions.

The dimensions of the model cells are specified in a discretization file of which there are three types:

- structured grid;
- discretization with vertices; and,
- unstructured grid.

Time is divided into stress periods which are further subdivided into time steps. Flows are calculated from the solution for head at each cell at the end of each time step. If the head convergence threshold is set too large, the inflow may not equal the outflow. This is described as a mass balance error. If the error is large then the solution is not acceptable and the convergence threshold, or possibly other model inputs, need to be adjusted to improve the model.

5.1 Consistent Units

MODFLOW requires that all data be input in consistent units. The length and time units used are input to MODFLOW. Thus, if the length unit is meters and the time unit is days, elevations of cells would be in meters, the length of stress periods would be in days, and pumping rates at wells would be in m^3/d. Use of inconsistent units must be avoided because the results of models with inconsistent units are not valid.

5.2 The Grid

In MODFLOW, the modeler must specify:

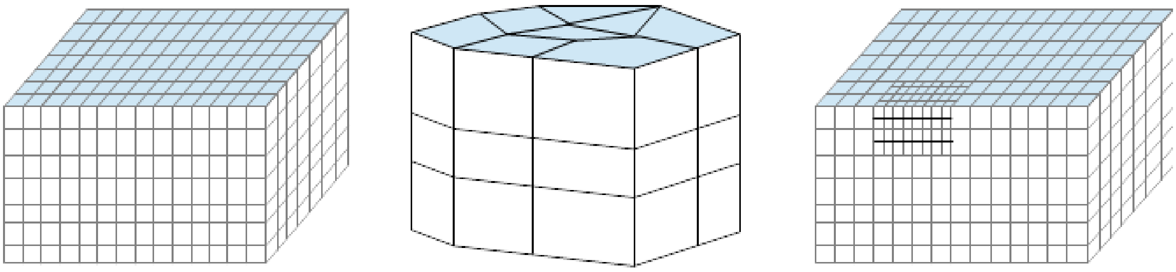
- the geometry of the system to be modeled;
- the initial head at every modeled location;
- the properties of the medium through which the groundwater flows; and
- the external features, such as rivers and wells, that control flow into or out of the groundwater system.

The initial heads can be heads from a previous MODFLOW simulation. If arbitrary values are used, such as land surface elevation or interpolated values from field data, then heads consistent with other model inputs need to be obtained by running a steady-state stress period before undertaking transient simulations. Interpolated values from a potentiometric map should not be used for the initial heads in a transient simulation because the solution will be adjusting to the inconsistency between the heads, properties, and boundary conditions as well as to the transient stresses, so the results will not be valid.

MODFLOW does not calculate the flow at every possible location within the model domain. That would be impossible because there are an infinite number of locations within the model volume. Instead, MODFLOW 6 divides the model domain into polyhedrons (called cells). Heads are calculated at the centers of cells. It is critical that the modeler carefully consider the level of grid complexity required for the system. MODFLOW 6 provides three options for spatial discretization. Discretization is division of the aquifer system into smaller volumes (model grid cells).

Three types of discretization are available:

- *Structured grid:* where the cells are rectangular prisms (Figure 29);
- *Discretization by vertices:* where the grid is made of polygons in the horizontal plane and this polygonal grid is repeated in one or more layers (Figure 29b); and,
- *Unstructured grid:* where cells are polygonal in plan view, but the cells may not be organized into layers or the cells in different layers have different sizes (Figure 29c).



a) b) c)

Figure 29 - Grid types: a) structured with layers, rows, and columns; b) discretization by vertices having continuous layers across the grid but without continuous rows and columns; and, c) unstructured without continuous layers, rows, and/or columns.

5.2.1 Structured Grids

In structured grids, the cells are arranged in a grid (Figure 30a). In the plan view, all the cells in each column have the same width and the cells in each row have the same length. In cross section, however, the vertical height of each cell can be different (Figure 30b). Flow is only calculated between cells that lie adjacent to one another. However, adjacency is determined only by the column, row, and layer numbers of the cells; the elevations of the cells have nothing to do with whether the cells are considered adjacent or not. Thus, in Figure 30b, cells A and B are considered adjacent to one another even though they do not appear to share an edge because they are both in layer 2 but cells B and C are not adjacent because they are in different layers. Cells in MODFLOW have a single top and bottom elevation, as presented in Figure 30b. The alternative representation in Figure 30c makes it easier to identify the connections among cells within a layer at the expense of misrepresenting the top and bottom of the cells as sloped rather than flat.

For an illustration of cell connections within a layer, see Box 3⤵.

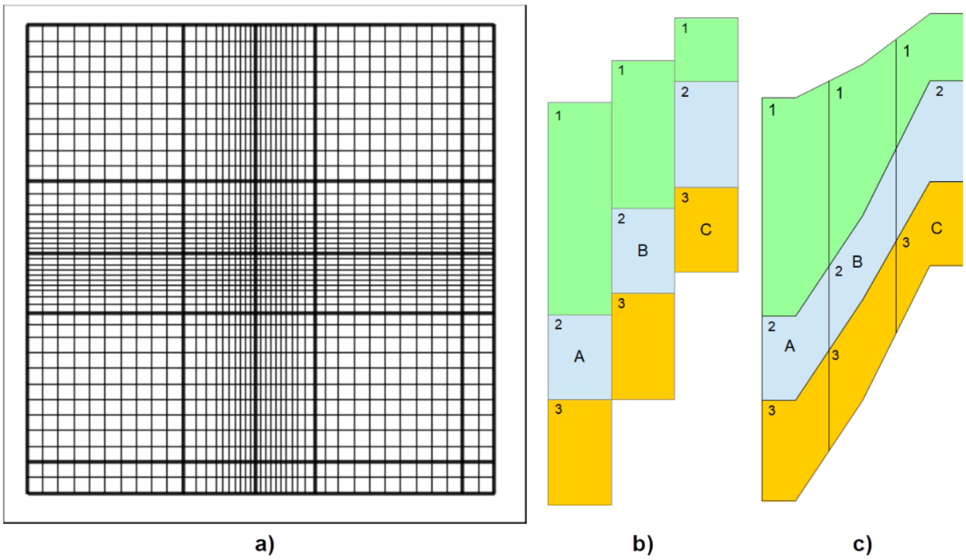


Figure 30 - Example structured grid: a) plan view; b) side view of three columns in a grid; and c) alternative representation of a grid.

Cells are typically identified by their layer, row, and column numbers in that order.

- Layers are numbered from the top down vertically;
- rows are numbered from the top of the page down; and
- columns are numbered from left to right.

The directions of the row and column numbers are conventions. They can be reversed without affecting the calculations. The layer numbering, however, must be from top to bottom vertically. With these conventions we can show the grid numbered with layer, row, and column numbers (Figure 31)

(1,1,1)	(1,1,2)	(1,1,3)	(1,1,4)	(1,1,5)
(1,2,1)	(1,2,2)	(1,2,3)	(1,2,4)	(1,2,5)
(1,3,1)	(1,3,2)	(1,3,3)	(1,3,4)	(1,3,5)
(1,4,1)	(1,4,2)	(1,4,3)	(1,4,4)	(1,4,5)
(1,5,1)	(1,5,2)	(1,5,3)	(1,5,4)	(1,5,5)

Figure 31 - Structured Grid with cell identifications

(Layer, Row, Column).

The widths of the rows and columns in the grid can vary (Figure 30a); smaller cells can be used where a more detailed solution is required, and larger ones can be used elsewhere. Because MODFLOW is solving the system of equations for groundwater level (head), detailed solution is necessary where the water level surface has larger changes in gradients, such as near streams or around wells; these may be areas where smaller cells are necessary for solution accuracy. Additionally, smaller cells may be needed where large changes in hydraulic conductivity occur, as these areas may have steep hydraulic gradients between cells. Using variable row and column widths can reduce model execution time as compared to a model with a uniformly small cell size. However, if neighboring rows or columns are assigned significantly different widths, accuracy of the model can be adversely affected. Typically, the ratio of widths of adjacent rows and columns should be kept less than 1.5:1 (Anderson et al., 2015).

5.2.2 Discretization by Vertices

When discretization is accomplished using vertices, the grid can be irregular (polygonal) in the horizontal plane, but this grid must be the same in all layers (Figure 29b). Cells are identified by layer number and the sequence number in the polygonal grid. Neighboring cells in overlying and underlying layers can be identified automatically because they have the same sequence numbers. Neighboring cells in the horizontal plane are identified by cell faces with shared vertex numbers. Flow can occur between neighboring cells, but the neighboring cells must share a cell face. The discretization file includes the vertices that define the boundary of each cell. The concept of neighboring cells can be explored by undertaking Exercise 2⤵.

5.2.3 Unstructured Grids

With unstructured grids, the cells are identified by sequence number alone. All cell neighbors must be identified by the modeler. The modeler must also specify whether a neighboring cell is a horizontal or vertical neighbor. Neighboring cells must share a cell face.

The horizontal connections between cells can be defined differently in unstructured grids in MODFLOW 6 than in structured grids or discretization with vertices. For example, in the model cross section shown in Figure 32, if the grid is structured or discretized with vertices, cell 8 would have cell 9 as a horizontal neighbor. However, if the grid is unstructured, cell 8 might also have cells 3 and 6 as horizontal neighbors if the user chooses to identify them as such. When this approach is used, the grid is referred to as a vertically staggered grid (Panday et al., 2013; Langevin et al., 2017). Vertically staggered grids can be helpful for simulating faults.

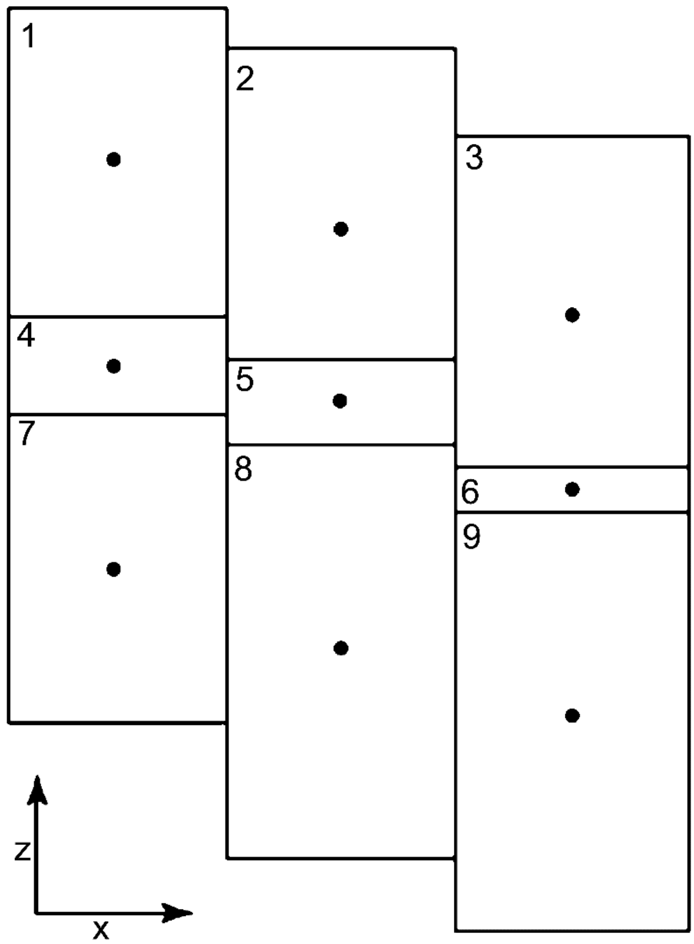


Figure 32 - Diagram of cross section through a model grid used to demonstrate the concept of a traditional horizontal connection and a vertically staggered horizontal connection. For a traditional MODFLOW representation (structured grid), a cell connects to only one other cell in a given horizontal direction (for example, cell 8 is connected on its right face only to cell 9). For a vertically staggered connection (unstructured grid), a cell may connect to more than one cell in a horizontal direction (for example, cell 8 is connected on its right face to cells 3, 6, and 9). Vertically staggered horizontal connections can be specified for the unstructured Discretization (DISU) Package, but not for the structured Discretization (DIS) or the Discretization by Vertices (DISV) Packages." Modified from Langevin et al. (2017).

5.2.4 Grid Design

The grid design affects the accuracy of the solution. In MODFLOW, the grid ideally should be designed so that a line drawn from the center of a cell to a connected cell intersects the boundary between the cells at a right angle. The connecting line should intersect the boundary line at the center of the boundary line. The cells should be convex. Deviations from these criteria can adversely affect the model accuracy. Structured grids meet these criteria automatically. Grids made up entirely of equilateral triangles or regular hexagons also meet these criteria.

In many cases, the grid may not meet the ideal requirements. If the violations are small, some modelers may elect to ignore the problems and accept the possibility that they degrade the accuracy of the solution. However, MODFLOW 6 provides two options for increasing the accuracy of the solution when the grid is less than ideal.

1. The Ghost-Node Correction package (Panday et al., 2013)
2. The XT3D Option in the Node Property Flow package (Provost et al., 2017)

These options are described in Section 7.3⤵.

Experienced modelers frequently prefer to use structured grids with uniform cell sizes if the area is not large and the total number of model cells is manageable. Such grids tend to reduce rounding errors of the linear system of equations formed by the finite-volume approximation to the groundwater flow equation for the area that the model represents. Other types of grids are used for large (over 10,000 km^2) areas with many aquifer layers when reduction of the total number of cells is desirable in order to reduce the time required to run the model while still having sufficiently small cells in areas where the water table has higher gradients.

The grid in MODFLOW can be oriented at any angle. MODFLOW does not need and may not record the location of the grid in space, the grid angle, or the geographic projection used for the grid. The modeler should record all this information in the documentation of the model, as comments in one or more of the input files, or as a separate document. However, it is good practice to keep the hydrogeologic data used to create the model data sets in a GIS data system that is independent of the model input files. It is good practice for modelers to select one geospatial map projection to use for all the model data sets. ModelMuse like most other GUIs allows for GIS files to be imported.

5.2.5 Active Cells

MODFLOW can calculate the head at every cell in a rectangular grid. However, the modeler usually designates some cells as inactive because groundwater systems are rarely rectangular in shape, the layers may not all have the same areal extent, and there may be areas within the system perimeter where the system does not exist (e.g., outcrops of bedrock in an unconsolidated alluvial aquifer). MODFLOW does not include inactive cells in its calculations, that is, heads are only calculated for active cells and flow is not calculated for the faces between active and inactive cells.

The boundary between the active and inactive cells is a no-flow boundary. By default, the edges of the grid (including the top and bottom surfaces) are also no-flow boundaries. Water crosses the edge of the grid only if the modeler specifies a boundary condition at the edge of the grid that allows for flow. Boundary conditions can also be specified in the interior of the grid. For example, the modeler can specify an injection or withdrawal well in the grid interior to add or remove water from groundwater in the model. Rivers, wells, recharge, and evapotranspiration are all examples of boundary conditions that allow flow into or out of the system.

One of the MODFLOW 6 input files contains an array called the IDOMAIN array. The IDOMAIN array has one value for each cell. The IDOMAIN array defines the active and inactive cells in the model. Cells with positive values in the IDOMAIN array are active. Those with values of zero are inactive. If a cell has a negative value in the IDOMAIN array, it is treated as a vertical pass-through cell. In vertical pass-through cells, flow is routed directly between overlying and underlying active cells; the vertical pass-through cell itself is not included in the calculation. If an Unstructured grid is used, IDOMAIN is optional. If IDOMAIN is not included in an unstructured grid, every cell is assumed to be active.

5.3 Stress Periods

Just as space in MODFLOW is divided into cells of finite size, time is broken up into discrete time steps. Heads are only calculated at the end of each time step. The time steps are grouped together into "stress periods." Within a stress period, the "stresses" on the system, such as the pumping rates at wells, are usually treated as constant. It is possible to specify changes in stress at every time step using "time series" as discussed in Section 10. The lengths of time steps within a stress period cannot be set to arbitrary values. Instead, the modeler specifies the length of each stress period, the number of time steps within each stress period, and a "multiplier." The multiplier must be a number greater than or equal to one. If the multiplier is one, each time step within a stress period has the same length. If the multiplier is greater than one, the lengths of all the time steps except the first will be determined by multiplying the length of the previous time step by the multiplier. MODFLOW will determine the length of the first-time step so that the time step lengths add up to the length of the stress period.

Equation (1) shows how the length of the first-time step in a stress period is calculated.

$$\Delta t_1 = PERLEN \frac{TSMULT - 1}{TSMULT^{NSTP} - 1} \tag{1}$$

where:

Δt_1 = length of the first time-step

$PERLEN$ = length of the stress period

$TSMULT$ = time-step multiplier

$NSTP$ = number of time steps in the stress period

5.3.1 Time-Step Multipliers

Stresses on the system only change at the beginning of a stress period. A time-step multiplier greater than one is useful when there is a large change in stresses between periods. When the stresses change, there can be rapid and large changes in the heads, flow rates, and flow directions. If a large time step is used to represent a rapidly changing system, the new heads and flows calculated at the end of the time step may be inaccurate. For example, if a hole is dug into a groundwater system, inflow is rapid at first then slows as the surrounding heads decline. However, if this is simulated using a long initial time step, the calculation will not allow the model to account for the decline of the surrounding heads during that time step period, and more inflow will occur in the simulated time step than in the field setting. Adjustments to abrupt, large changes can be difficult to calculate numerically, but it is easier if the initial time steps are small. Later, after the system has adjusted to the changed stresses, adequate accuracy can be obtained with larger time steps. Because larger time steps allow the model to run faster, larger time steps are desirable as long as the accuracy is adequate.

Generally, time steps need to be smaller for smaller cell sizes. A rough estimate of the maximum allowable step length for a square cell can be determined by Equation (2) (Anderson et al., 2015, page 319).

$$\Delta t_i = \frac{Sa^2}{4T} = \frac{S_s b a^2}{4Kb} = \frac{S_s a^2}{4K} \tag{2}$$

where:

Δt_i = initial time step

S = storativity

T = transmissivity

S_S = specific storage

a = length of cell

K = hydraulic conductivity

b = cell thickness

However, this is only an estimate of an appropriate initial time step length. The only way to be sure this time step is sufficiently short, is to run the model with a shorter time-step length and evaluate whether the shorter step produces different heads and/or flows.

The level of inaccuracy resulting from any particular time discretization will vary depending on the position of the cell within the grid. Locations close to sources and sinks generally require smaller time steps to reduce errors to acceptable levels than is necessary at other locations. If there is significant drawdown near a no-flow boundary, locations close to the no-flow boundary may also require shorter time-step sizes.

It is common for the initial versions of a model to have time-step sizes that are too long. Groundwater models iterate to reach a solution and the modeler specifies a maximum number of iterations after which the simulation must terminate. If the model fails to converge after reaching the maximum number of iterations, one reason could be that the time steps are too long. On rare occasions, time steps that are too short can be a problem. In such models, the solver may converge in only a few iterations but the percent error in the mass balance (the difference between inflows and outflows) may be unacceptably large.

An example of how time-step size affects accuracy is provided in Box 4↴.

As an alternative to specifying the number of time steps and their size, the modeler can employ Adaptive Time Stepping in any stress period. With Adaptive Time Stepping, the modeler specifies an initial time step length, a minimum and maximum time step length and factors used to adjust the time step length. MODFLOW will use these to adjust the length of time steps so that the number of outer iterations required for the model to converge is neither too few nor too many. If the model fails to converge on any time step, MODFLOW will reduce the time step length and repeat the time step until either the model converges, or the time step length would need to be reduced below the minimum time step length.

5.3.2 Steady State Versus Transient

If the stresses on a system are unchanged for a sufficiently long time, the heads and flow will eventually stop changing. That is, the amount of water flowing into the system will match the amount leaving the system and heads and flows within the system will not change over time. Such a condition is called steady state. If the model system is changing over time, it is in a transient state.

It is useful to use a steady-state model or stress period to simulate the initial heads for a transient model and MODFLOW accommodates this. The modeler can designate the first stress period as a steady state stress period and other stress periods as transient stress periods. In a steady state stress period, the model needs only one time step, and the specified length of the stress period is irrelevant. For transient conditions, models typically use multiple time steps and often more than one stress period. Some aquifer systems may never be in steady state, for example a tidally affected aquifer.

6 Initial Conditions

One question that a MODFLOW modeler needs to confront is how to assign the initial heads for a model. For steady-state models, this usually isn't much of a problem. So long as the initial heads are not so low that they cause cells to be dry (i.e., head below bottom of the layer) at the start of the simulation, MODFLOW will typically be able to solve for the steady-state heads regardless of the initial heads. With transient models, having good initial heads is more important. One way is to start with pre-development conditions in a steady-state model where recharge and surface water features are included but wells, which are part of development, are not included. Because MODFLOW allows some stress periods to be steady-state and others to be transient, this can be accomplished by making the first stress period a steady-state stress period. Additional stress periods can be defined for transient conditions. A new stress period is defined each time the modeler wishes to change stresses such as specified flow rates or constant heads. MODFLOW 6 allows variation of pumping rates and some other transient data within a stress period by using time-series files described in Section 10.

In large-area models, seasonal effects can often be ignored. However, in smaller-area models, seasonal or other cyclical effects may dominate the aquifer response. In such cases, instead of starting with a steady-state stress period, the modeler can repeatedly use a series of stress periods representing the average cyclic conditions. For example, the cycles could be four changes per year representing seasonal cycles or four changes per day representing tidal fluctuations. When enough cycles have been simulated, the change from the beginning of one full cycle to the next should be minimal and the resulting heads from any point in the cycle can be used as the initial heads to start a transient simulation at that point in the cycle. In such models, once the cyclic equilibrium is established, it is prudent to simulate and calibrate to several cycles representing actual conditions before the prediction period. For example, if predictions are to be made after 2020 and calibration data are available from 2000 to 2020, then enough cycles should be simulated to reach cyclic equilibrium by 2000, then the model should be calibrated to observations between 2000 and 2020. Once the calibration is acceptable, the heads at the end of the simulation in 2020 can be used to start the predictive scenarios from 2020 onward.

Figure 33 illustrates how the year-to-year changes in modeled head can decrease after several seasonal cycles. It shows the simulated head at one cell for daily time steps over a period of five years. The model has 50 rows, 50 columns and 3 layers each 10 m thick. Specified heads of 25 were used in column 50 of layer 1. The first stress period is a steady-state stress period with a recharge rate of 1×10^{-5} m/day. After that, stress periods alternated between 90 and 275 days in length. The recharge rate during the shorter stress periods is 3.33×10^{-5} m/day and the recharge rate during the longer stress periods is 1×10^{-6} m/day. In this case, it takes two annual cycles before year-to-year change is minimal.

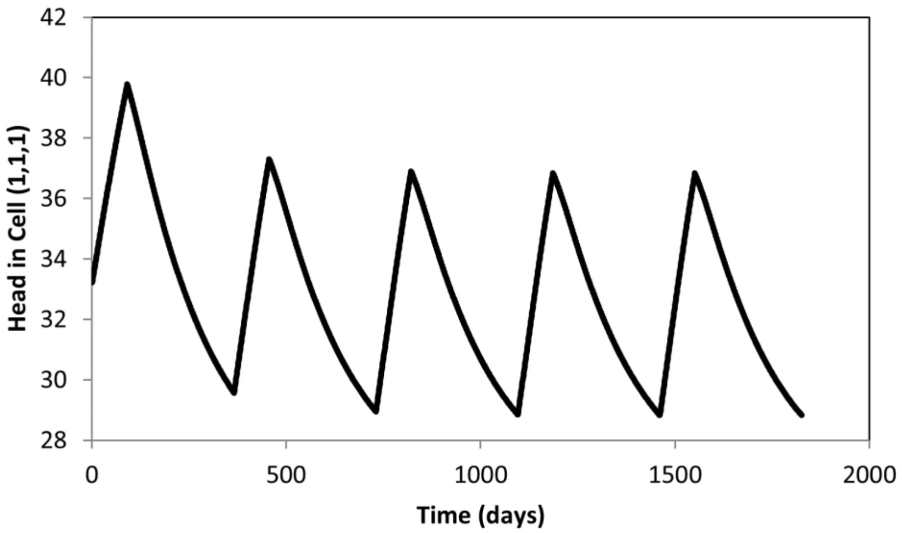


Figure 33 - Plot of head in a cyclic transient model illustrating that after a sufficient number of cycles, the changes between cycles is small. The number of cycles required depends on the hydraulic properties (i.e., diffusivity, which is transmissivity divided by storativity, and controls the rate at which a pressure wave propagates and dampens out in an aquifer), geometry, and location of boundary conditions within the groundwater system.

Some systems are never in a cyclic equilibrium. For them, another method for estimating the length of the warmup period must be used. The warmup period is the simulated time after which the heads no longer depend on the initial conditions. This time can be estimated by perturbing the initial conditions. The warmup period must be simulated prior to using head and flow observations for model calibration. The required length of warmup period is determined by developing transient stress period boundary condition datasets prior to the simulated period of interest. (Boundary conditions are discussed in detail in Section 8). Actions for estimation of the warmup time are as follows.

1. Create transient stresses and boundary conditions for the calibration simulation period.
2. Insert a copy of the data for the first transient stress period into the model but make it steady-state and make it be the first stress period.
3. Run the model and save the heads from that first steady-state stress period as well as the simulated values from the other stress periods.
4. Add several meters to the heads for the first stress period calculated in action 3 and use those for the initial heads in action 5. The amount to be added should be a few percent of the difference in head between the highest and lowest simulated heads in the model.
5. Run the model using the perturbed heads from action 4 as initial conditions and omitting the steady-state stress period.
6. Determine how long it takes for the difference in heads between the simulation in action 3 and the heads from the simulation of action 5 to be inconsequential (i.e., for

the heads to be very close because they may never match perfectly). To be completely clear, this difference being evaluated is between simulated heads starting with stress period 2 of the simulation in action 3 and simulated heads starting with stress period 1 in the simulation of action 5.

The length of simulated time required for the difference to be inconsequential (i.e., action 6) is the estimated warmup period because after that length of time, the simulated heads no longer depend on the initial conditions. The aquifer properties might be modified during calibration, so it is prudent for the modeler to prepare transient stress data sets for a longer period than this initial estimate and simulate these stress periods (the warmup period) prior to simulating the time period containing the observations for calibration. If a model already includes simulation of a relatively long period before any calibration observations are available or groundwater extractions occur, this warmup period is not required because the simulated time before groundwater extraction acts as the warmup period.

7 Representation of Properties of Geologic Materials

The properties of the geologic system play a critical role in determining the flow of groundwater. These properties are specified in two packages. Hydraulic conductivities and related properties are specified in the Node Property Flow package. It is used in all models. Specific yield and specific storage are specified in the Storage package. It is used in transient models. If the Storage package is included in a steady-state model, the values specified in it will be read but will not be used.

7.1 The Node Property Flow Package

In the Node Property Flow (NPF) package, the modeler specifies the hydraulic conductivity in three mutually perpendicular directions. By default, the directions of the three hydraulic conductivities correspond to the X, Y, and Z coordinate directions of the model grid. However, the user can specify a different orientation for every cell. This means that any cell can be specified to have anisotropy of hydraulic conductivity in a fully three-dimensional orientation that differs from the cartesian coordinates used for the model as a whole (Provost et al., 2017). To do this, the modeler specifies six variables. The first three variables are the hydraulic conductivities in three orthogonal directions. The other three variables are angles (in degrees) that specify the orientation of the hydraulic conductivity ellipsoid (Figure 34). If all three angles are zero, the three hydraulic conductivities will be oriented parallel to the X (red), Y (blue), and Z (green) axes respectively (Figure 34a). The first angle rotates the ellipsoid within the XY plane around the z-axis (Figure 34b). Positive values represent counterclockwise rotation when viewed from the positive end of the Z axis. The second angle rotates the ellipsoid around the Y axis of the ellipsoid out of the XY plane (Figure 34c). A positive value rotates the positive end of the X-axis above the XY plane. The third angle rotates the ellipsoid around the X or major axis of the ellipsoid along which the first hydraulic conductivity is oriented (Figure 34d). A positive value is a clockwise rotation when viewed looking down the positive end of the X or major axis toward the center of the ellipsoid (Figure 34d). The angles and hydraulic conductivities are used to calculate effective hydraulic conductivities between cells. The values of all three angles are combined to determine the final orientation of the ellipsoid (Figure 34e).

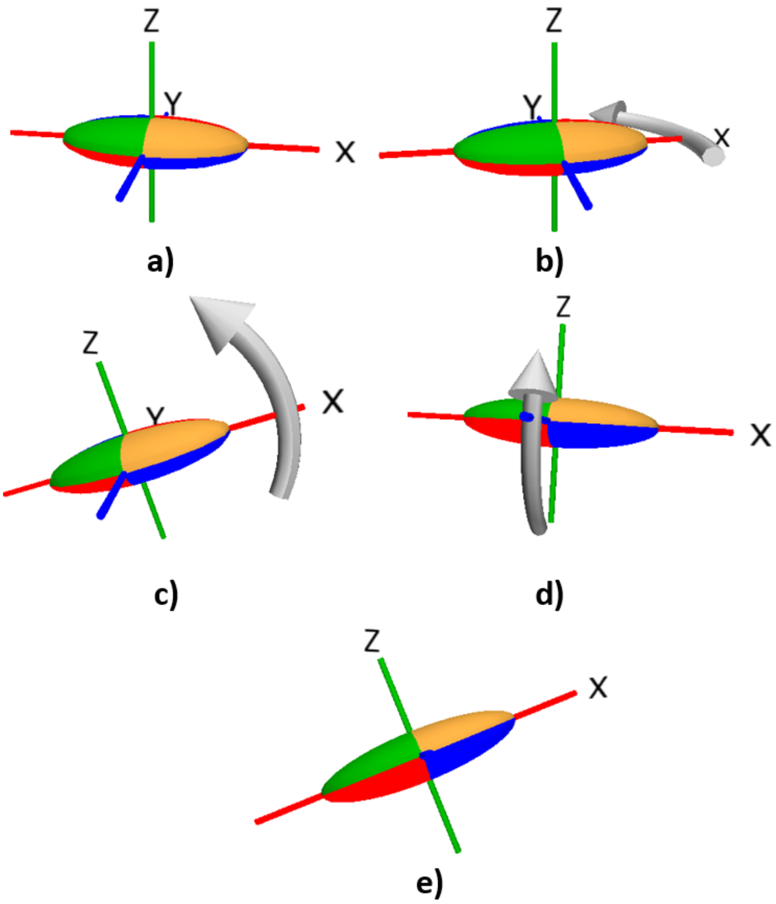


Figure 34 - Illustration of Angles in NPF package. a) Illustration of hydraulic conductivity ellipsoid with no rotation. b) Angle1 set to 20°. c) Angle2 set to 20°. d) Angle3 set to 20°. e) All angles set to 20°. The axes shown are the axes of the hydraulic conductivity ellipsoid.

The second and third hydraulic conductivities can be specified either directly or as anisotropies relative to the first hydraulic conductivity. Using anisotropy can be useful if automated parameter estimation is used to calibrate the model and it is anticipated that all three components of hydraulic conductivity are controlled by the same factors. For example, suppose all three angles are equal to zero and it is expected that the horizontal hydraulic conductivity is the same in all directions. In that case, it would be useful to use a horizontal anisotropy of 1. Similarly, if it is expected that the vertical hydraulic conductivity has a constant ratio to the horizontal hydraulic conductivity, vertical anisotropy could be specified instead of vertical hydraulic conductivity.

To calculate the flow between two cells, MODFLOW uses the length, width, and height of the cells (provided in the Discretization file); the heads in the two cells; the hydraulic conductivity between the two cells; and the cell status regarding confined/unconfined condition of the cells. Note that the hydraulic conductivity used in the calculation is not the hydraulic conductivity of the cells themselves but of the volume between the centers of the cells. MODFLOW uses the properties of the cells to calculate an average hydraulic conductivity between the cells.

Calculating an average value for the properties of two cells might seem straightforward, but it is complicated. Consider a very simple case with two cells that are the same size: one with a high hydraulic conductivity and the other with a low hydraulic conductivity as shown in Figure 35 (units = m/s).

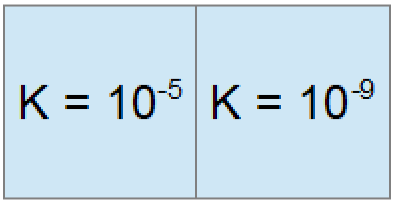


Figure 35 - Hydraulic conductivities (m/s) used to calculate conductance.

The arithmetic mean of these two hydraulic conductivities is 5.0005×10^{-6} m/s. This is a high hydraulic conductivity that does not properly represent the resistance of the low hydraulic conductivity cell on the right which would greatly impede the flow of water between the cells. In fact, for groundwater flow, which follows the same principles as the flow of electrical current, the average hydraulic conductivity is accurately represented by using the distance-weighted harmonic mean. The details of calculating the harmonic mean are not important here, but it is useful to recognize that a harmonic mean weights small values more heavily than large values. In the example above, the harmonic mean is 1.9998×10^{-9} m/s. The options for both grid design and calculation of flow conductance terms between cell centroids are explained in more detail by Panday and others (2013), Langevin and others (2017), and Provost and others (2017).

Each cell in a MODFLOW model can be specified as either confined or convertible using the ICELLTYPE data set in the Node Property Flow package. The groundwater flow computations are different for convertible cells than confined cells. One of the factors controlling the groundwater flow rate is the cross-sectional area through which the flow occurs, and this depends on the thickness of the flow zone. For confined conditions, that cross-sectional area is the cell height times its width. For unconfined conditions in convertible cells, the cross-sectional area depends on the height of the water table. Thus, for convertible cells, MODFLOW checks whether the head is above or below the top of the cell to determine whether the cell is in a confined or unconfined condition, then calculates the flow area based on the saturated thickness (i.e., cell height for the confined condition, or the difference between the head and the bottom elevation of the cell for an unconfined condition) and cell width (Figure 36). When convertible layers are used this requires an iterative solution because the cell-by-cell conductance is a function of head (the solution of the linear system of equations).

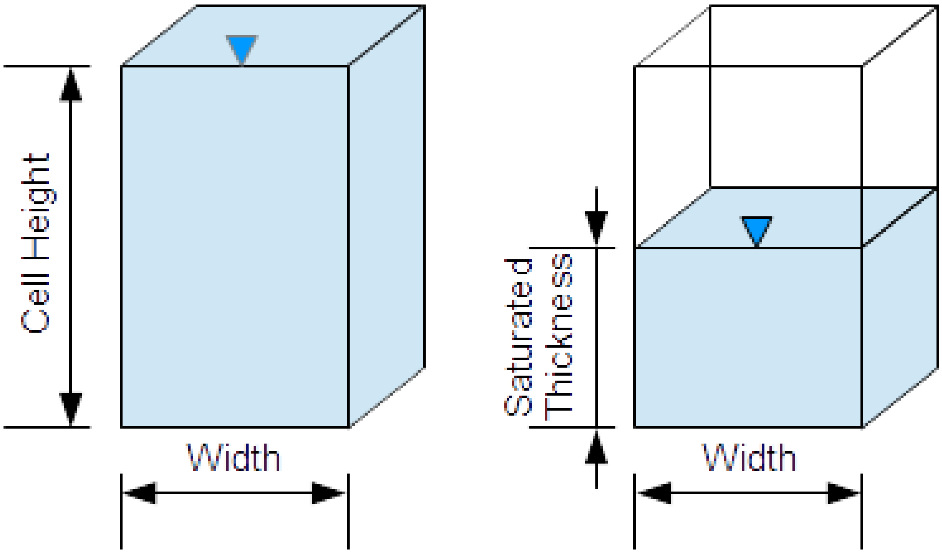


Figure 36 - Saturated thickness used to calculate conductance in confined (left) and unconfined (right) cells.

Often modelers classify all layers beneath the top layer as confined for faster run times. The fact that a cell is specified as confined does not mean that the simulated head in the cell will never drop below the top of the cell, it only means that the transmissivity and storativity do not change based on water level in the cell. MODFLOW can simulate heads above, within, or below the bottom of a confined cell without numerical problems. However, this can cause the model to poorly represent the physical system and MODFLOW does not check for heads below the bottom of a confined layer. Thus, the modeler needs be mindful of this possibility and, if such a situation occurs, reject the results as unrealistic and improve the model to avoid the situation.

The NPF package can use either of two different methods for calculating the flow between neighboring cells. In the standard formulation, convertible cells can convert to dry and become inactive if the head in the cell is below the bottom of the cell. If the head in neighboring cells become high enough, the dry cells can become wet again and convert back to active cells. This wetting and drying can cause the model to be unstable and fail to converge. In the Newton-Raphson formulation, cells never become inactive, rather flow is calculated based on properties of the upstream cell. Thus, the Newton-Raphson formulation avoids the problem of wetting and drying that plagues the standard formulation.

7.2 The Storage Package

The storage properties of each cell are specified in the MODFLOW Storage package. It is also used to specify which stress periods are steady-state and which are transient. Storage properties are only required for transient simulations so if no transient stress periods are defined, the storage properties are not used, and the Storage package may be omitted. If the Storage package is included in a steady-state model, the storage properties will be read but not used. For a transient model, the user must specify either a specific storage or a storage coefficient (the product of specific storage and cell thickness) for every cell. If there are any convertible cells in the model, the user must also specify the specific yield for every cell in the model.

During a MODFLOW simulation, first the flows between neighboring cells are calculated, then a new head is calculated for each active cell. If the head in a convertible cell is above the top elevation of the cell, it is fully saturated, and the new head is calculated using specific storage of the cell. If the cell is not fully saturated, the new head is calculated using a combination of the specific yield and the specific storage of the cell.

7.3 Ghost Node Correction Package and XT3D Option

The Ghost Node Correction Package (Langevin et al., 2017) is one of two options in MODFLOW 6 for correcting errors arising from non-ideal grid geometries. The other is the XT3D option. These two options cannot be used together in the same model.

Both options require additional calculations and thus can increase the time required to run the model. The XT3D option (Provost et al., 2017) slows down the model more than the Ghost-Node Correction package, but it can be easier to apply and is thought to give better accuracy.

7.3.1 The Ghost Node Correction Package

In the Ghost-Node Correction (GNC) package, the modeler defines "ghost nodes" whose heads are determined by interpolation from surrounding nodes. For example, in computing the flow between nodes A and D in the grid illustrated in Figure 37, the ghost node B is defined, and its head is determined by interpolation between nodes A and C. Because ghost node B is directly across from node D and the line connecting B and D is at the center of the boundary of the cell containing D, flow computed using the ghost node is more accurate than flow computed directly between A and D. The modeler must specify how the interpolation is to be performed by specifying weights for each cell that contributes to the head of the ghost node. While the GNC package can be used in any of the three MODFLOW 6 grid types, there is no obvious reason why it would be needed within a single structured grid. However, if a simulation contains more than one flow model with separate model grids, ghost nodes can be used in linking two separate structured grids that are discretized differently.

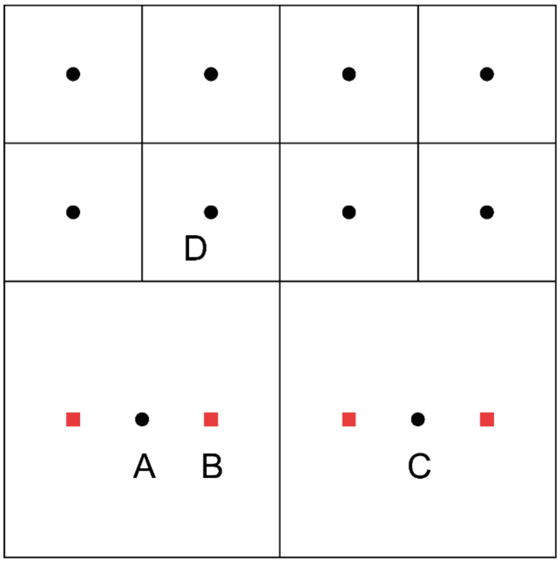


Figure 37 - A quadtree refined grid showing locations of nodes and ghost nodes. The nodes are black circles at the centers of cells. The ghost nodes are red squares. The head of ghost node B can be determined by interpolating between nodes A and C.

One type of grid for use with MODFLOW 6 is generated by modifying a structured grid in which a cell is divided into four cells of equal size as shown in Figure 38. If desired, the refined cells can be subdivided further in the same manner ad infinitum. This is known as quadtree refinement. The utility program GRIDGEN (Lien et al., 2015) can be used to convert a structured MODFLOW grid into a quadtree refined grid. Part of the output from the program is the information used in constructing the GNC package input. ModelMuse (Winston, 2019) uses a slightly different method for constructing a quadtree refined grid and the corresponding ghost node information.

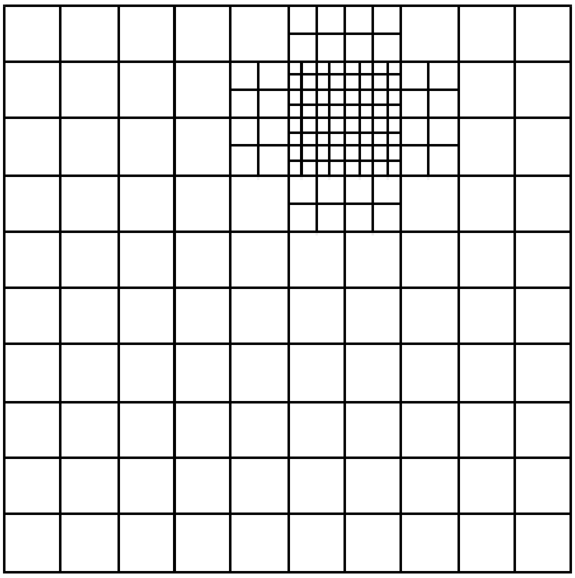


Figure 38 - Example of a Quadtree Refined Grid (Lien et al., 2015).

There remains the difficult question of how to assign the interpolation weights to the nodes that contribute to the head of the ghost node. This problem is discussed by Panday and others (2013). One possibility is to assume that head changes linearly between the nodes that contribute to the head in the ghost node. In Figure 37, this would mean that for ghost node B, node A would be assigned a weight of 0.75 and node C would be assigned a weight of 0.25. Another option was used in MODFLOW-LGR (Mehl and Hill, 2006, 2007) in which the weights automatically assigned to ghost nodes were based not only on the distances involved but also on the transmissivities of the cells. The problem of assigning weights becomes even more difficult when the cells are not rectangular. In such cases, it may be best to use the XT3D option rather than ghost nodes.

7.3.2 The XT3D Option

With the XT3D option (Provost et al., 2017), the calculation of the flow between two cells involves the heads of all the neighbors of those cells. In Figure 39, the flow between the cells whose centers are marked with black circles needs to be calculated. In addition to using the heads in those two cells, the calculation also involves the heads in all the cells whose centers are marked with blue squares. In the calculation, head gradients are calculated at the location on the boundary between the cells marked by the red star. Two gradients are calculated: one on each side of the boundary. The gradients are calculated by interpolating among the cell and all its neighbors. Because the principal of continuity must be honored, flow across the boundary must be the same on each side of the boundary. This constraint, used together with the head gradients, allows flow across the boundary to be calculated.

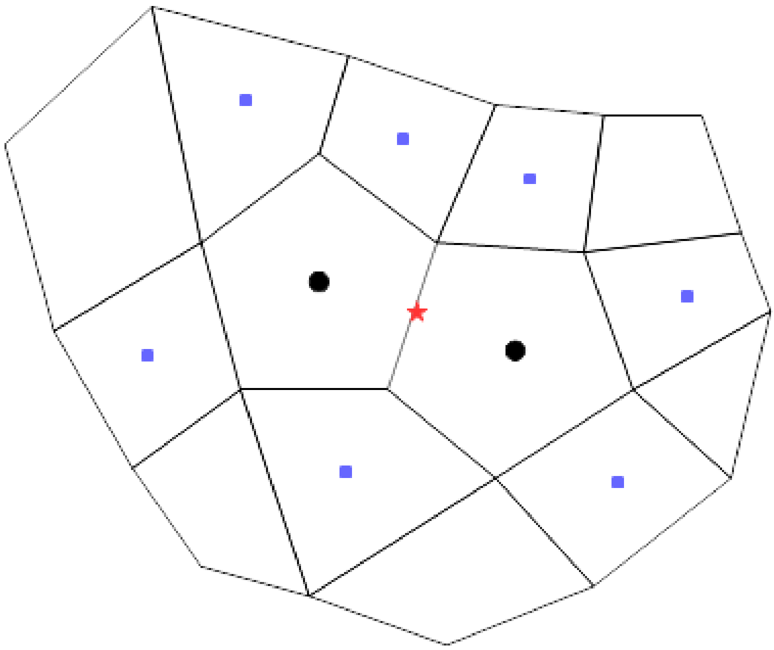


Figure 39 - Illustration of the cells involved in the calculation of flows between two neighboring cells.

When using the XT3D option, the user must specify six values for defining hydraulic conductivity at each cell. These data define an ellipsoid of hydraulic conductivity and its orientation (Figure 34). Three of the data values define the maximum, middle, and minimum hydraulic conductivity. The other three pieces of data are angles (in degrees) that determine the directions of the three axes as explained in Section 7.1.

8 Boundary Conditions

While the properties of the groundwater medium control how groundwater flows, it is the boundary conditions that are the motive force driving groundwater flow. The boundaries are where water enters and leaves the groundwater system (including places where the flow rate is zero, i.e., no-flow boundaries). The boundaries can be categorized into three types: specified head, specified flow, and head-dependent flow boundaries. A special case of the specified flow boundary is the no-flow boundary which is simply a specified flow boundary at which the flow rate is zero.

Not every cell in the model will be a boundary cell. Typically, the number of boundary cells is much smaller than the number of active cells in the model. The input for most boundary conditions is a list of cells along with the properties at those cells that control flow through the boundaries. With most boundary conditions, more than one boundary condition of the same or different type can be applied in the same cell. For example, you can have two wells in the same cell and that cell may also receive recharge from the surface and interact with a river reach. However, for cells that have a specified head boundary applied, only one specified head boundary can be applied to a cell for a stress period, as described in the next section. Other types of boundary conditions applied to a specified head cell will be ignored. In general, flow boundary conditions like recharge and wells do not cause numerical problems when multiple conditions are applied at the same cell as the flows are summed together. Problems with model convergence can occur if several head-dependent flow boundary conditions are applied at the same cell, especially if the specified heads of each head-dependent flow boundary condition differ greatly. Additionally, two adjacent head-dependent boundary cells can cause convergence issues if the heads are substantially different and the head-dependent conductance terms are large. This is discussed in Section 8.3.

8.1 Specified Heads: The Constant Head Package

In MODFLOW 6, the only way to define a specified head boundary is with the Constant Head (CHD) package. The name is something of a misnomer because it allows the modeler to specify heads that change over time. In versions of MODFLOW prior to MODFLOW 6, its official name was the Time-Variant Specified Head package but informally, it was commonly called the constant head package. Unlike the other boundary condition packages, the same cell cannot be listed in the CHD package more than once in the same stress period. After a CHD boundary has been defined, it can be converted to an active cell by not including it as a constant head cell in a subsequent stress period. Time-series files can be associated with specified head boundaries. Time-series files provide a mechanism for changing the specified heads within a stress period rather than at the beginning of a new stress period. A time-series file consists of a list of times along with one or more values associated with those times. They are described in more detail in Section 10.

Some modelers recommend that specified head boundaries never be used. That seems extreme to this author but there is potential danger associated with using specified head boundaries, because the amount of flow through them is unlimited. The flow may exceed the volume of flow that is realistically available from the area beyond the boundary. The modeler is responsible for ensuring that this does not occur.

Some boundaries are a natural fit for use of a constant head boundary. For example, for all practical purposes, groundwater flow from a small aquifer to the ocean will not change sea level in any significant way, and the ocean would be able to supply as much water as might flow into the aquifer, so the ocean can be represented as CHD boundary (either as an average value or with periodic fluctuations representing tidal rise and fall). Another example would be a reservoir in which humans control the reservoir level.

A constant head boundary allows the magnitude of inflow to be determined based on the time-changing heads calculated within a model, whereas the alternative of specifying the time-varying flux through the boundary is difficult to determine a priori. However, it is important to check the magnitude of flow at all constant head boundaries to confirm that they are realistic.

Readers who would like to get hands-on experience working with a MODFLOW model that uses the CHD package are invited to explore Exercise 3↴ and Exercise 4↴.

8.2 Specified Flows

MODFLOW 6 has two packages for specified flow boundaries: the Well package and the Recharge package. By default, the edge of the grid and the boundary between active and inactive cells are no-flow boundaries. No-flow boundaries are a special form of a specified flow boundary.

8.2.1 The Well Package

In MODFLOW models, the Well package is one of the most used packages. In it, the modeler gives a list of cells for each stress period and the pumping rate for each of them. If there is more than one well in a cell, the cell can be listed several times, or the rates can be summed and entered as one value. Positive values of pumping represent water injected into the aquifer and negative values represent water extracted from the aquifer. In most cases, wells are used to provide a source of water for human use (such as irrigation or drinking water) so usually the pumping rates are negative. If a cell containing a well converts to dry, the well will become inactive.

The Well package can be used to represent any specified volumetric rate of flow into or out of the groundwater system, not just flow to and from wells. For example, it might be convenient to have a local model of a small area within a regional groundwater flow model. The flow between the regional and local model could be represented with the well package.

There are several processes that can occur in a well that are not represented by the Well package in MODFLOW. For example, a well may be pumping from several aquifers simultaneously or, if the well is not being pumped but is open to several aquifers, there may be flow between the aquifers through the well. These and other effects can be modeled with the MultiAquifer Well package as discussed in Section 12.2.

MODFLOW 6 can use multiple Well package input files for one simulation. That is, one Well package might represent specified flows from a regional groundwater flow system and another Well package might represent wells in the interior of the model. The Well package has an option to gradually reduce well withdrawal (i.e., increase negative values of flow rate) to zero as head in the cell approaches the bottom of the cell (Figure 40). This option is only applied to convertible cells. If the pumping rate in any well is reduced, the reduction is printed either to the listing file or to a separate output file. This option enhances model stability by making it less likely that a cell will go dry.

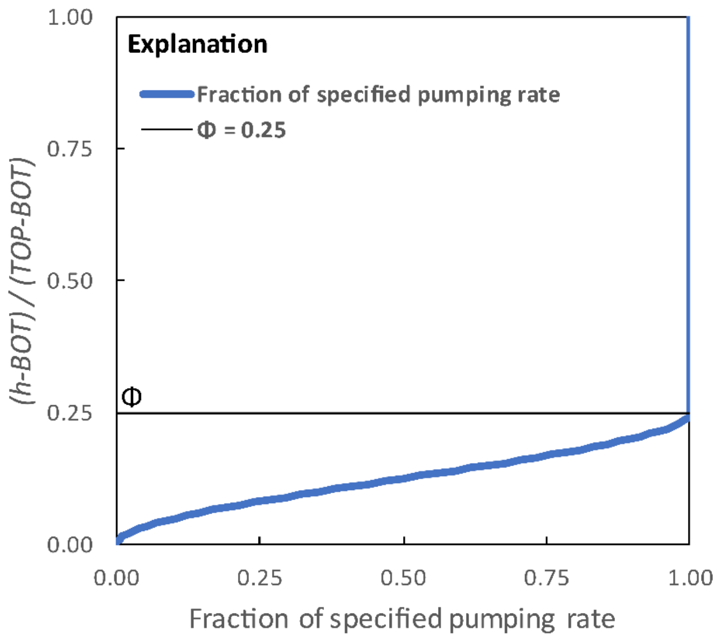


Figure 40 - Graph showing the cubic function used to smoothly reduce specified groundwater withdrawal rates to zero when a cell dewaters, $\Phi = 0.25$. (Modified from Langevin et al. (2017) as modified from Niswonger et al. (2011)). The flow reduction occurs when the saturated cell thickness is less than a user-defined percentage (Φ) of the cell thickness. In this diagram, Φ has been set to an unusually large value so that the effect on the flow rate is easily seen.

The modeler is advised to check for any wells that have had their pumping rates automatically adjusted. If the reduced pumping occurs in wells that represent flow rates measured in the field, the model is not properly representing the groundwater system, or the pumping rates would not have been reduced. If the pumping rates represent future estimated pumping rates, the reductions may mean that there will not be adequate water supply to meet future demands. In such a case (assuming the model is correct), either a new water supply needs to be developed or future water use reduced. Developing a new water supply might mean installing a well in a deeper aquifer, importing water from elsewhere or constructing a desalination plant. Reducing water use might mean removing irrigated farmland from production. Such changes may need to be represented in the model.

8.2.2 The Recharge Package

An important concept to understand about groundwater recharge is that it is not the same as precipitation. Recharge is water that reaches the water table from above. You can think of recharge as precipitation, minus interception, minus runoff, minus evapotranspiration from the unsaturated zone. Recharge can also result from flow out of surface water features, such as rivers, agricultural irrigation, or spreading basins, into the groundwater system.

In MODFLOW, recharge is a specified flux boundary (like wells). However, instead of specifying a volumetric rate (L^3/T) for each cell, the modeler specifies a rate that has units of length over time L/T for each vertical column of cells. MODFLOW multiplies this rate by the area of the cell to calculate the volumetric rate. In MODFLOW 6, recharge can be specified either as an array for all the cells in a layer or as a list of cells. If a list of cells is used, recharge can be applied to any cell in the model. If an array is used, the modeler has three possible choices about where to apply recharge:

- the top layer of the model;
- a specified layer for each vertical column of cells; or
- the top active cell in each vertical column of cells.

If a list of cells is specified, the user can specify an option in the Recharge input file to have the recharge applied to an underlying cell if the cell to which the recharge was applied becomes inactive.

MODFLOW does not apply recharge to specified head cells, even if those cells are included in an array specifying the cells where recharge is applied. If a cell for which recharge is defined in the input converts to dry, the recharge will not be applied to the model unless the third option above is selected and an active cell exists below the one that went dry.

Positive values of recharge represent flow into the aquifer and negative values represent flow out of the aquifer. Recharge values can be negative to represent a process other than recharge or to represent a system where evaporation occurs directly from the water table. However, MODFLOW has another package that better represents the process of water evaporating directly from the water table (the Evapotranspiration EVT Package which is discussed in Section 8.4).

MODFLOW 6 allows the user to specify multiple Recharge Package input files in the same model.

8.3 Head-Dependent Boundaries: Drain, River, and General Head Boundaries

Groundwater and surface water frequently interact with one another. Discharge of groundwater into streams, for example, is a major source of stream baseflow. Rivers and streams can also be recharge zones if the head in the underlying aquifer is lower than the head in the stream. In other situations, groundwater can discharge at locations where there is no possibility of re-infiltration. Artificial drains are one example of such a situation. Artificial drains are common in some areas. For example, in the United States, by the mid-1980s only about 103 million acres remained of the approximately 221 million acres of wetlands present in the 1600s (Dahl and Allord, 1996). Drainage of wetlands for agriculture or other development is a major factor in the elimination of wetlands.

The Drain and River packages are two of the simpler packages in MODFLOW for simulating interactions between groundwater and surface water. The General-Head Boundary package is similar in its operation to the Drain and River packages and is even simpler, so the General-Head Boundary package is presented first.

In the General-Head Boundary package, the modeler specifies three items for each general-head boundary cell. These are: identification of the cell containing the boundary (e.g., layer, row, and column for a structured grid); the boundary head (e.g., a surface water elevation); and a conductance that represents the resistance between the groundwater system and the boundary. The conductance is related to the hydraulic conductivity in Darcy's Law which is shown in Equation (3).

$$Q = -KA\frac{dh}{dl} \tag{3}$$

where:

Q = discharge rate (L^3/T)

K = hydraulic conductivity of the resistive material between the boundary head and the grid cell (L/T)

A = cross sectional area perpendicular to the flow between the boundary head and the grid cell (L^2)

h = head

l = thickness of the resistive material between the boundary head and the grid cell (L)

dh/dl = head gradient, the derivative of head with distance

In essence the conductance represents the combination of all the terms on the right-hand side of Equation (3) except dh. The dh term is determined by MODFLOW as the difference in head between the head in the cell and the head specified by the modeler for the general-head boundary. MODFLOW uses a finite volume approximation, so dh/dl is replaced by $\Delta h/\Delta l$ and conductance is defined as $KA/\Delta l$.

The cross-sectional area may be as large as the entire cell area if the feature occupies the entire cell. A feature such as a lake may overly many cells of a model and some of them would be completely inundated by the lake thus the area would be equal to the cell area. If the general-head boundary represents a feature entirely contained in a grid cell, perhaps a pond, then the area would be smaller than the cell area and equal to the area of the pond. General-head boundaries need not be on the top surface of a cell but might represent a feature to the side or underneath a cell. The general head boundary can also be used to represent flow from, or to, another aquifer across the boundaries of the model.

To determine the discharge (or recharge) rate (QB) at the general-head boundary, MODFLOW multiplies the conductance by the difference in head between the head in the general-head boundary and the head in the cell where the boundary is located (red line, Figure 41).

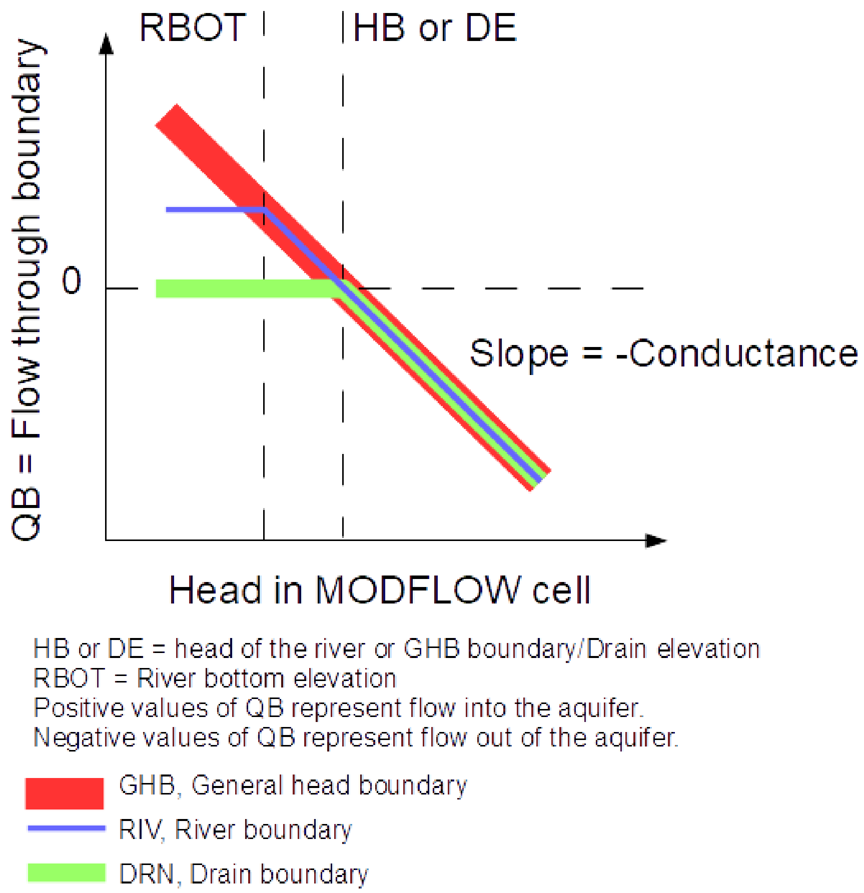


Figure 41 - Relationship between head in a cell connected to a boundary and flow into or out of the cell through the boundary for a general head boundary, river boundary, and drain boundary.

Unlike some of the other boundary conditions, there is no limit on the amount of water that can enter and leave the aquifer through a general-head boundary. Thus, if the conductance or head gradient are too high, an unrealistically high volume of flow may occur through the boundary. It is the responsibility of the modeler to detect unreasonable simulated flows and avoid them by revising the model.

The drain boundary is like the general head boundary with the drain elevation replacing the boundary head and the conductance representing the silt (or other low hydraulic conductivity material) that lines the bottom of the drain. However, with drains, if the head in a cell is below the drain elevation, there is no flow from the drain to the aquifer (green line, Figure 41). The conceptual representation of a drain is, unlike a river, that of a channel that does not convey surface flow except to carry groundwater outflow away from the system. Consequently, water does not flow from a drain feature into a MODFLOW grid.

In some cases, modelers have placed drains in every cell of the top surface of a model to represent water exiting the system (e.g., by evaporation or surface drainage) when the groundwater head reaches the surface. If the conductances of the drains are high enough, this prevents the simulated water table from rising above the ground surface. In such models, the modeler can map the amount and location of discharge through the drains to assess whether the outflow is realistic. If the discharge is excessive or occurs in unrealistic locations, it may be that the recharge rate for the model is too high, or the hydraulic conductivity is too low. In such cases, the modeler may adjust these or other aspects of the model to make the model more representative of the field system.

A river boundary is like a drain with two exceptions: 1) it has two elevations associated with it: the stage (i.e., head) of the river and the elevation of the river bottom; and 2) water can flow into the groundwater system from the river. That is, depending on the relative heads of the river and the cell to which it is connected, flow can be into or out of the groundwater system. When the head in the cell connected to the river boundary is below the river bottom, water flows out of the river into the aquifer at a rate that is independent of the head in the cell. The rate of flow is calculated using Darcy’s law based on the stage of the river; the elevation of the riverbed (where the pressure is assumed to be atmospheric so the head is equal to the elevation); and, the conductance of the riverbed which is envisioned as having a lower hydraulic conductivity than the aquifer. Water from the river is assumed to flow away from the bottom of the riverbed into the unsaturated zone (i.e., the zone beneath the bottom of the riverbed and above the water table) as fast as water is supplied from the base of the riverbed. The flow rate under these circumstances does not depend on the elevation of the water table (horizontal part of blue line in Figure 41, Figure 42b). When the head in the cell is above the bottom of the river bed, the flow into or out of the groundwater system depends on the stage of the river, the head in the cell, and the conductance of the riverbed (sloped part of blue line in Figure 41, Figure 42a).

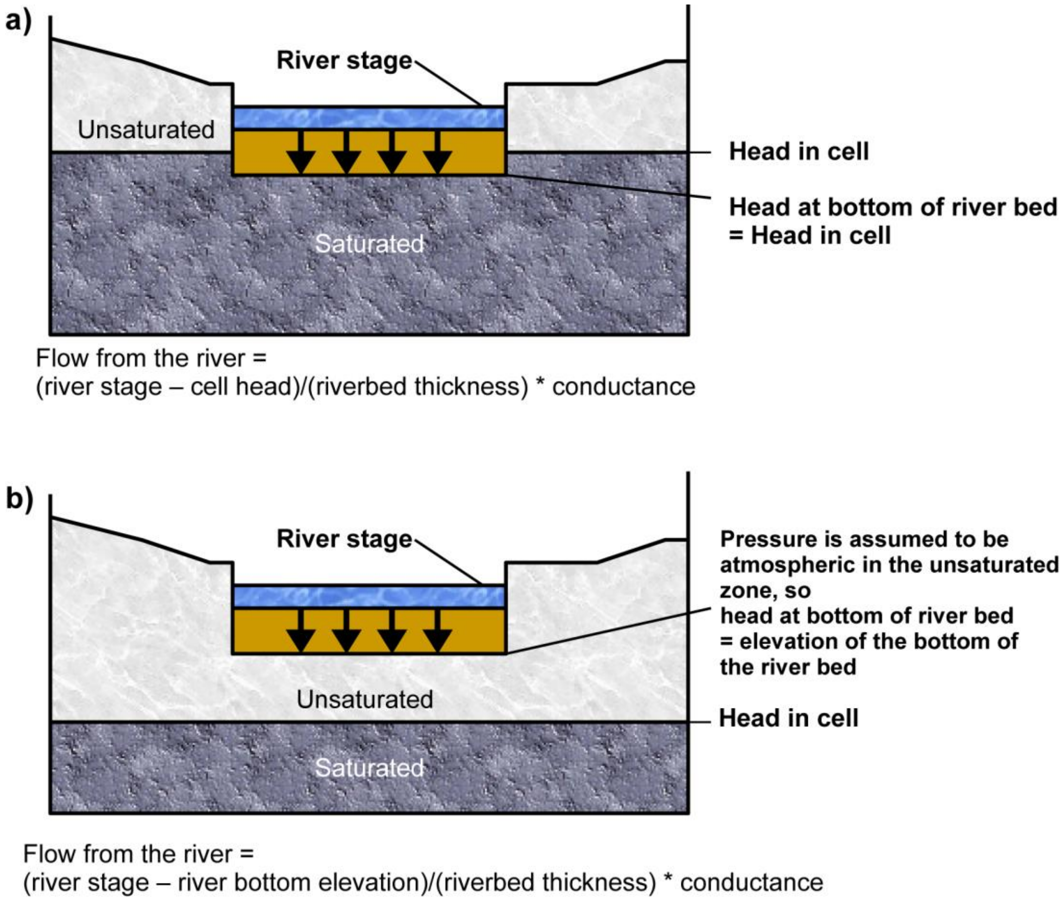


Figure 42 - Cross sections showing the relation between head at the bottom of the riverbed layer and head in the cell. Head in the cell is equal to the water-table elevation (Modified from McDonald and Harbaugh, 1988). The flow rate is lower in a) relative to b) because of the smaller head difference, resulting in a lower gradient. If the cell head is higher than the river stage water flows out of the groundwater system to the river.

The abrupt break in slope of the functions shown in Figure 41 for the drain and river are referred to as discontinuities or nonlinearities. In the early stages of calibrating a model, it can be helpful to use the general head boundary package at locations where the river or drain packages will eventually be used because the linear response of the general head boundary package makes it easier to calibrate the model. The boundaries can be changed to drain or river boundaries and the conductances used in the general-head boundaries can be used as estimates of the conductances in the drain or river boundaries. The general head boundary can also be used when there is flow from or to another aquifer across the boundaries of the model or to simulate lakes that maintain a constant water level.

In MODFLOW 6, multiple copies of the drain, river, and general head boundary package input files can be used in the same model.

If the cell containing a drain, river, or general-head boundary converts to dry in a convertible cell, then flow between the groundwater system and the boundary will be zero.

8.3.1 Cell Size and Conductance

Calculating conductance from the geometry of the boundary and hydraulic conductivity of the material separating the grid cell from the boundary head might appear to imply that if the grid is modified, the conductance can be determined based only on the changed geometry. This could be the case if a feature specified on a cell covers the entire cell and the rows and columns of the cell are re-discretized, but is not strictly true for several reasons so the modeler must give careful thought to the conductance values if the grid is changed.

For example, consider a case where the conductance has been adjusted to properly simulate observed flow between the cell and a boundary feature in the process of calibrating the model. The head in the cell depends on cell size because the head in the cell is an average head for the cell volume and the flow to the boundary depends on that head. If the cell is further discretized in the lateral direction the average head calculated in the cell with the boundary may differ and so will the combined flow from all the cells. This can be even more pronounced if the layers are re-discretized, particularly if there is a vertical gradient. Because of this, the conductance required to generate a specific volume of discharge through a boundary can change if the model is re-discretized (Mehl and Hill, 2010). In short, conductances represent equivalent properties that allow the model to properly represent the system given its discretization and may differ from the measured values of the boundary materials such as a riverbed.

If the model cell sizes are large, then there could be large river stage differences between adjacent cells along a river and the conductance term could also be large if calculated from river length and width. This can be the source of slow convergence and sometimes can be overcome with a finer model grid or by reducing the head-dependent conductance term.

8.4 The Evapotranspiration Package

The main package in MODFLOW 6 for simulating evapotranspiration (evaporation plus plant transpiration) is the Evapotranspiration (EVT) package. The Unsaturated Zone Flow package can also simulate evapotranspiration but is less frequently used. The EVT package does not simulate all evapotranspiration, because much of it occurs at the surface or from the unsaturated zone. Instead, the EVT package simulates evapotranspiration of groundwater at or below the water table. The EVT package assigns an extinction depth at which no evapotranspiration will occur if the water table is below that depth. For example, if the extinction depth is defined as 1 m beneath the evaporation surface elevation (often specified as the ground surface elevation), evapotranspiration will be nonzero when the water table is less than 1 m below the surface and zero when it is deeper. Also, the EVT package specifies a maximum evapotranspiration rate when the head in the cell is at or above the evaporation surface elevation because the rate will be limited by atmospheric conditions. For each evapotranspiration cell, the user must specify the maximum evapotranspiration rate, the evapotranspiration surface, and the evapotranspiration extinction depth.

Beginners frequently find the evapotranspiration extinction depth confusing. It is the distance from the evapotranspiration surface to the elevation at which evapotranspiration is zero. The evapotranspiration extinction depth is not an elevation. In Figure 43, for example, the evapotranspiration extinction depth has the same value everywhere, but the evapotranspiration surface elevation varies from place to place.

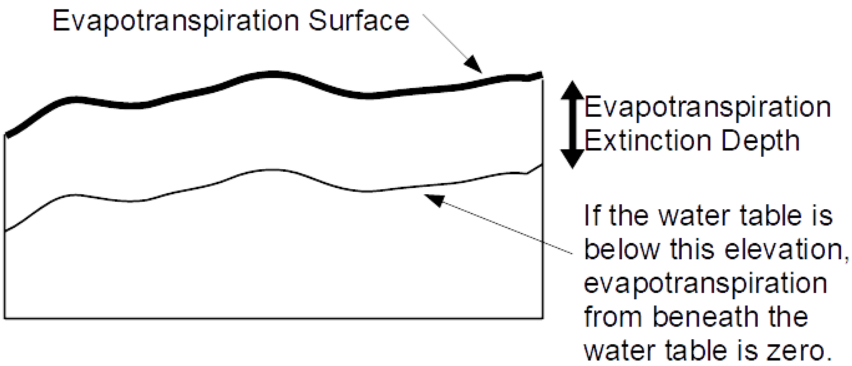


Figure 43 - Diagram illustrating a constant extinction depth in the EVT package combined with a varying evapotranspiration surface.

In the EVT package, the maximum rate is given as a positive value with units of length over time (LT^{-1}). The head in a cell is used to compute the rate for that cell. MODFLOW multiplies that rate by the length and width of the cell to compute a volumetric rate.

In the simplest usage of the EVT package, the evapotranspiration rate is linearly related to head between the surface and extinction elevations (Figure 44a). However, the EVT package allows for a non-linear relationship between head and evapotranspiration rate within that range (Figure 44b). To define a non-linear relationship between head and evapotranspiration rate, the modeler defines a series of line segments relating the evapotranspiration rate to the head in the cell. If this option is used, the modeler defines two sets of fractions. One set represents fractions of the evapotranspiration extinction depth and the other represents fractions of the maximum evapotranspiration rate. Using a nonlinear relationship can be advantageous for locations with a mixture of plants, some of which have shallow roots and others deep roots, because when the cell head is beneath the shallow roots, only the deep-rooted plants will transpire water from beneath the water table. At higher heads, all plants will transpire water from beneath the water table (Figure 44).

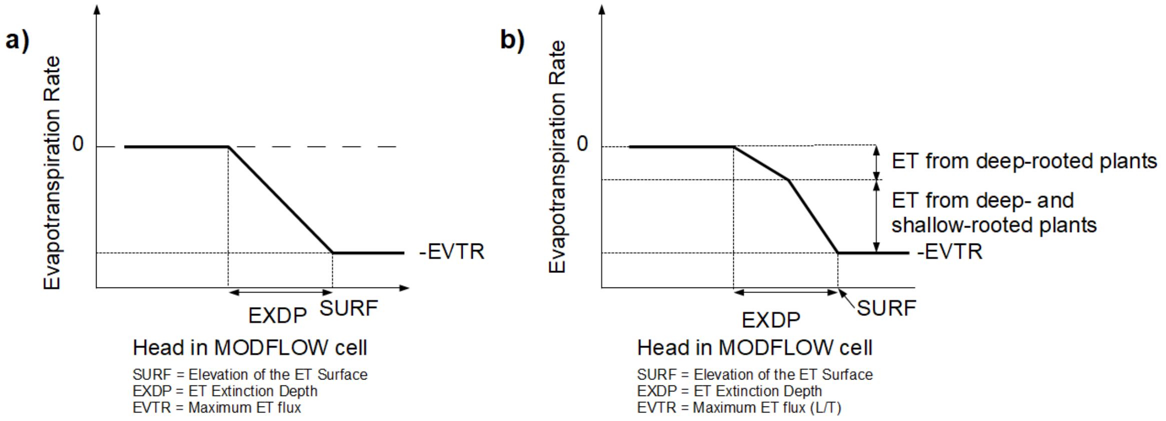


Figure 44 - Relationship between head and evapotranspiration rate in the Evapotranspiration package a) simplest representation; and b) nonlinear representation.

In MODFLOW 6, it is possible to apply evapotranspiration to any or all cells in the model because MODFLOW 6 allows the user to specify evapotranspiration either as an array of cells or a list of cells. The layer to which evapotranspiration is applied can also be assigned in the same way as with recharge: to the top layer, to a specified layer, or to the top active layer. MODFLOW 6 also allows more than one EVT package input file in the same model. As with recharge, if the cell to which an evapotranspiration boundary is assigned converts to dry, there is an option for the evapotranspiration to be assigned to a lower cell. If there is no active lower cell, the rate will be zero.

8.5 Opportunities to Learn More by Working with MODFLOW Models

Exercise 5⤵ provides an opportunity for hands-on work with MODFLOW specified flow and head dependent flux packages.

Model size is often limited by the time required to solve the model. For that reason, it is common to construct models using boundary conditions that are not present in the physical system. Exercise 6⤵ explores the influence of model boundaries on the simulated flow system and introduces simulation of transient conditions.

Exercise 7⤵ provides an opportunity to use a MODFLOW companion code, MODPATH, to track paths of flow through a MODFLOW model.

9 Other Packages

9.1 The Iterative Model Solution Package

MODFLOW models must include a solver package that solves the groundwater flow equation. In MODFLOW 6, the only solver package is the Iterative Model Solution. (IMS) package. In it, the user can specify *Simple*, *Moderate*, or *Complex* options and it can support either the standard or Newton-Raphson linearization methods (Langevin and other, 2017). For a discussion of an earlier version of the IMS package, see the documentation for MODFLOW-USG where it is called the Sparse Matrix Solver. The discussion there provides additional details that are also applicable to MODFLOW 6 (Panday and others, 2013). The most detailed discussion of the IMS input variables is in the file mf6io.pdf which is provided in the “doc” directory of the MODFLOW 6 distribution file (USGS MODFLOW 6 Development Team, 2022).

The solver solves the equations iteratively; it first finds an approximate solution to the equations and then uses the approximate solution in the equations to get a better solution. This is repeated until the solution meets criteria specified by the modeler. There are two types of iterations: outer and inner. In each outer iteration, the equations are linearized using the most recent approximate solution. Then the linearized equations are approximated in a series of inner iterations to obtain a better solution before starting a new outer iteration.

The most important variables in the solver package are the convergence criteria variables. There are three such criteria: one for the outer iterations and two for the inner iterations. The criterion for the outer iterations is a head closure criterion. With it, the maximum change in head between two outer iterations in any cell must be less than the value of the head closure criterion for the solution to be accepted. The inner iterations have a head closure criterion, but they also have a residual closure criterion. The head closure criterion for the inner iterations should be less than the head closure criterion for the outer iterations. Typically, it is an order of magnitude less than the head closure criterion for the outer iterations. The other criterion for the inner iterations is a residual criterion. The residual criterion is related to the budget error (residual) in the cells. Before a solution can be accepted, the budget error in the cells must be small enough to satisfy the criterion. There are three options for the residual criterion. If the STRICT option is used, the largest residual in any cell must be less than the criterion. If the L2NORM_RCLOSE option is used, the L2-Norm of the residuals must be less than or equal the residual closure criterion. The L2-Norm is calculated as the square root of the mean of the squared residual in all the active cells. If the RELATIVE_RCLOSE option is used, the change in the L2-Norm between iterations is compared to the residual closure criterion to determine whether the solution has converged rather than comparing the absolute value of the L2-Norm. By default, the STRICT option is used.

Sometimes it is worthwhile to compare the solutions with different values of the closure criteria to see how much the solution changes. If reducing the closure criteria results in a substantial change in the solution, the larger closure criteria are too large. Another indication that the closure criteria are too large is if the overall percent discrepancy in the water budget is too large. Discrepancies between 0.5 and 1 percent are of concern and greater than 1 percent are too large. However, while a large percent error indicates the convergence criteria are too large, a small percent error does not mean the convergence criteria are small enough. It is a good idea to reduce the percent discrepancy in the overall water budget as much as possible given the trade-off with computation time.

Meeting the head and residual closure criteria does not ensure that the error in computed heads and flows are less than the closure criteria. In some models, the effects of the boundary conditions can take many iterations to propagate over the entire grid. In such cases, the errors have the potential to be substantially larger than the closure criteria. For this reason, it is best to keep the closure criteria small. For example, the models illustrated in Box 4 all met the closure criteria, but the calculated heads vary substantially among them especially at the edge of the grid.

The IMS package has options to print the progress of the solution process in each iteration. The data are written to CSV (comma-separated values) files. This makes it easy to open the files and plot the data with spreadsheet programs (Figure 45). Values related to the solution are printed for each iteration. If the model has not converged, it can be worthwhile to look at these results. If the absolute value of the maximum head change in the outer iterations is decreasing with each iteration, convergence might be achieved by allowing a larger number of outer iterations. On the other hand, if the absolute value of the maximum head change in the outer iterations is not decreasing with each iteration, increasing the allowable number of outer iterations probably will not help.

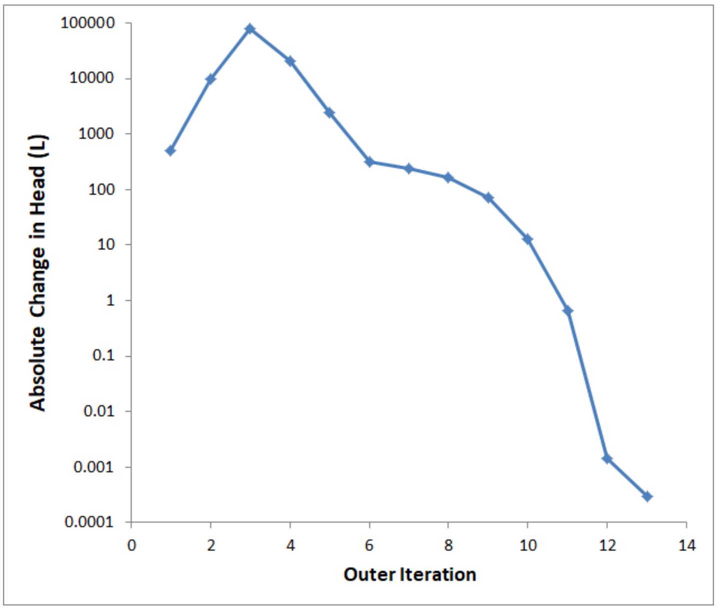


Figure 45 - Plot of convergence data from a model that successfully converged.

If a model does not converge, the modeler can look in the output from the solver to identify which cells are frequently not meeting the criteria. Sometimes examining the model configuration in the vicinity of such cells can reveal a problem with the model. If the wetting option is active, it is a good idea to evaluate whether the cells that are not meeting the criteria are repeatedly switching between wet and dry status. Increasing the wetting threshold for those cells may help with convergence.

The IMS package has many other options that can be used to change how the solver operates. Sometimes the maximum change in head or residual will oscillate between positive and negative values without ever meeting the convergence criteria. There are several options that may reduce and eliminate the oscillation. First, there is a complexity option that can be set to “Simple,” “Moderate,” or “Complex.” These options affect many of the default values applied to the other options as shown in Table 1 and Table 2. In addition, the modeler can specify values for any of the variables in Table 1 and Table 2 to override the default values. Descriptions of these variables are given in the file mf6io.pdf in the MODFLOW 6 distribution.

Table 1 - Nonlinear IMS variables in MODFLOW 6 version 6.3

Nonlinear variable	default/simple	moderate	complex
OUTER_DVCLOSE	0.001	0.01	0.1
OUTER_MAXIMUM	25	50	100
UNDER_RELAXATION	NONE	DBD	DBD
UNDER_RELAXATION_THETA	1.0	0.9	0.8
UNDER_RELAXATION_KAPPA	0.0	0.0001	0.0001
UNDER_RELAXATION_GAMMA	0.0	0.0	0.0
UNDER_RELAXATION_MOMENTUM	0.0	0.0	0.0
BACKTRACKING_NUMBER	0	0	20
BACKTRACKING_TOLERANCE	0.0	0.0	1.05
BACKTRACKING_REDUCTION_FACTOR	0.0	0.0	0.1
BACKTRACKING_RESIDUAL_LIMIT	0.0	0.0	0.002

Table 2 - Linear IMS Variables in MODFLOW 6 version 6.3

Linear variable	default/simple	moderate	complex
INNER_MAXIMUM	50	100	500
INNER_DVCLOSE	0.001	0.01	0.1
INNER_RCLOSE	0.1	0.1	0.1
RCLOSE_OPTION	STRICT	STRICT	STRICT
LINEAR_ACCELERATION	CG	BICGSTAB	BICGSTAB
RELAXATION_FACTOR	0.0	0.97	0.0
PRECONDITIONER_LEVELS	0	0	5
PRECONDITIONER_DROP_TOLERANCE	0.0	0.0	0.0001
NUMBER_ORTHOGONALIZATIONS	0	0	2
SCALING_METHOD	NONE	NONE	NONE
REORDERING_METHOD	NONE	NONE	NONE

If the model has convergence problems and changing the complexity option does not help, it may be that the model has features that are causing the convergence issue such as unreasonably high conductances, sharp contrasts in hydraulic conductivity within layers, poorly specified boundary conditions, or time steps that are too long. If none of these are applicable, then adjusting the under-relaxation variables may improve the solution process to reach convergence. With under-relaxation, the change in head between iterations is reduced which can facilitate convergence.

Backtracking has a similar impact, but there may be several backtracking iterations in which the changes in head are reduced in order to satisfy a flow-residual criterion and this may require a lot of computation time.

The ideal solution variable values are problem-dependent and difficult to predict. Typically, acceptable values are found by trial and error.

9.2 The Output Control Option

Output Control is used to specify the time steps for which to print heads to the listing file, save heads to an external file, print a budget summary to the listing file, or save the budget terms for each cell to an external file. For each stress period, the user can specify the time steps at which the calculated heads or flows are printed or saved. There is a trade-off between the size of output files and the frequency with which output is printed and saved. The simulation and evaluation of results can take much longer if a lot of output is requested. Consequently, it is best to consider which information is important to the project analysis and limit the output to include only that information.

The printed budget summary is short and includes the percent discrepancy for the model so frequent printing of the budget summary is not burdensome and can be useful.

9.3 The Streamflow Routing Package

The Streamflow Routing (SFR) package in MODFLOW 6 is like the River package. In both packages, flow between the groundwater system and the boundary can be in either direction. Like the River package, the rate of flow is controlled by the conductance of the boundary and the difference in head between the stream stage and the groundwater cell. However, unlike the River package, flow into a stream does not disappear from the model. Instead, it is routed to the next downstream stream cell and may change the stage of that stream boundary. Each stream cell keeps track of how much water is flowing through it. The flow in the stream will limit flow from the stream to the groundwater if the volume of inflow to the groundwater cell exceeds the volume available in the stream. The potential flow from the stream to the groundwater is calculated using heads and conductance. If all the available water flows into the groundwater system, no water will be routed downstream to the next stream boundary.

To route the flow properly, the connections between stream reaches must be specified (Figure 46). For each reach, the user specifies both the upstream and downstream reaches. The upstream reaches are specified with positive numbers and the downstream reaches are specified with negative numbers. If a reach has more than one downstream reach, the user must specify how the flow will be divided between the reaches. One of the downstream reaches will be considered the main flow channel and the others will be considered diversions. There are three options for specifying the amount of flow that is diverted. It can be a fraction of the total flow, an amount more than some specified value, or the amount diverted can be specified and the remainder will flow to the main channel.

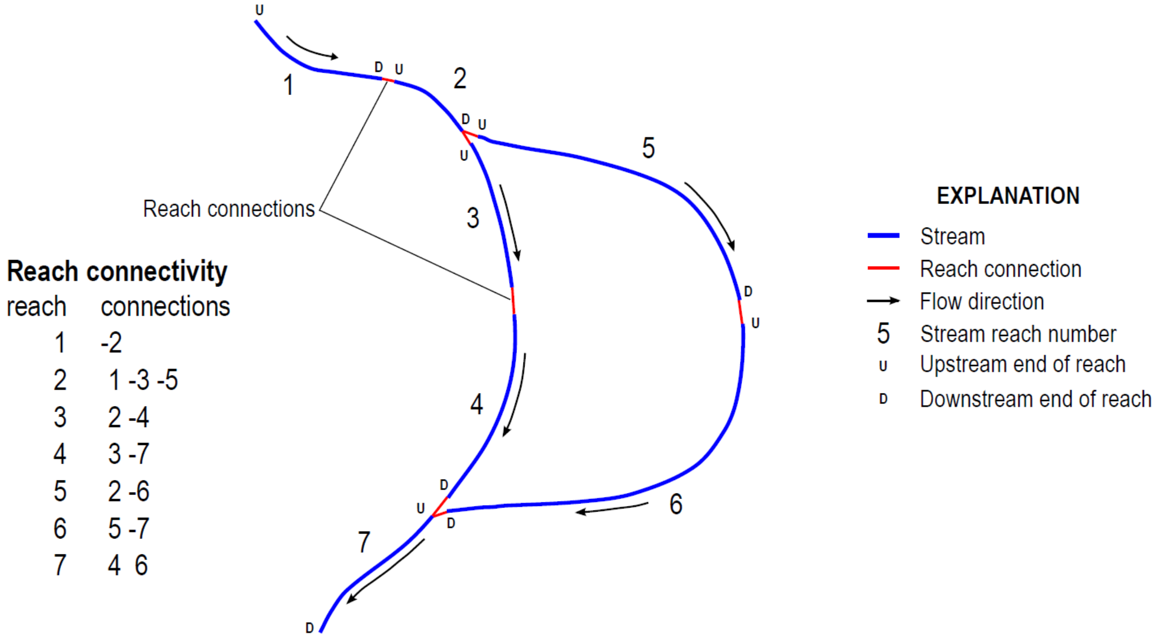


Figure 46 - Simple stream network having seven reaches with a junction having two reaches, a confluence of two reaches, and the resulting reach connectivity. All downstream connections for a reach must include the reach as an upstream connection. Downstream connections for a reach are denoted with a negative reach number (From USGS MODFLOW 6 Development Team, 2022).

Unlike the River package, conductance in the Streamflow Routing package is not specified directly. Instead, the modeler specifies the length and width of the stream in a cell along with the hydraulic conductivity and thickness of the stream bed. If desired, the modeler can specify a cross section of the stream channel. From these, MODFLOW calculates the stream conductance.

Another difference between the Streamflow Routing package and the River package is that the Streamflow Routing package has an option for having the head in the stream calculated instead of being specified by the modeler. The Streamflow Routing package uses Manning's equation to calculate the stream stage from the flow rate. For MODFLOW to make this calculation, the modeler specifies two additional inputs: the stream roughness and the stream gradient.

The modeler can specify the amount of rainfall and runoff entering a stream reach. The rainfall rate is multiplied by the reach length and width to determine the volumetric rate of precipitation on the reach. Runoff is specified as a volumetric rate. Water can also leave a stream reach by evaporation. The evaporation rate provided by the modeler is multiplied by the reach length and width to determine the volumetric rate of evaporation.

9.4 The Horizontal Flow Barrier Package

The Horizontal Flow Barrier package is used to simulate barriers to flow that are much smaller than the grid cells. For example, slurry walls or volcanic dikes could be simulated with the Horizontal Flow Barrier package. For each barrier, the modeler specifies two cells in the same layer on opposite sides of the barrier and the hydraulic characteristic of the barrier (Figure 47). The hydraulic characteristic is the hydraulic conductivity of the barrier divided by its thickness in the direction perpendicular to flow across the barrier. The hydraulic characteristic is used by MODFLOW to modify the conductance between the cells.

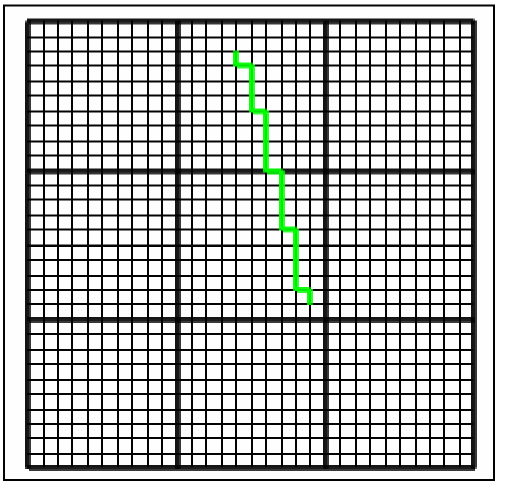


Figure 47 - Illustration of horizontal flow barriers in green along the interfaces between cells.

10 Time-Variable Input

Typically, the time variable input for MODFLOW 6 is specified using stress periods and remains constant for the entire stress period. However, for some inputs, it is possible to specify time variable input in a different way so that the input might change in every time step. This is done using either Time-Series files or Time-Array-Series files.

10.1 Time-Series Files

In MODFLOW 6, time-series files provide a general mechanism for specifying transient data that is independent of the stress periods of the model. In a time-series file, there is a list of times and one or more series of real-number values associated with those times. Each series of values is given a name. That name can be included in one of the other model input files. For example, in the constant head (CHD) package, a time-series file could be used to give a different specified head at every time step. The times specified in a time-series file will be interpolated to the appropriate model time step. One of three different interpolation methods can be used: “STEPWISE,” “LINEAREND,” or “LINEAR.” Data for an example time series with times from 0 to 6 is shown in Table 3. If there were more than one time series in the file, an additional column of data would be present for each additional series. Table 4 shows the value that would be used for the time step with each of the different interpolation methods for a time step extending from time = 0 to time = 5. Figure 48 illustrates the calculations graphically.

Table 3 - Example data for a time-series file

Time (days)	Associated value
0	0
1	0
2	1000
6	1200

Table 4 - Value interpolated by optional interpolation methods for time = 5 given values in Table 3

	Interpolated value
Stepwise	600
Linear	745
LinearEnd	1150

- With STEPWISE, it is assumed that values change abruptly at each new time, instead of gradually between times (Figure 48a). A time-weighted average of the values in the series during the time step is used. In this case, the time-weighted average is [(0×2 days) + (1000×3 days)]/5 days = 600.

  If LINEAREND is used, MODFLOW linearly interpolates between the time values surrounding the time at the end of the time step and uses that interpolated value for the entire time step (Figure 48b). In this case, the final value is (1200 - 1000) × (3 days / 4 days) + 1000 = 1150.
- If LINEAR is used with the same times and values, MODFLOW interpolates linearly between the values associated with times 0, 1, 2, and 6 and constructs a

time-weighted average value for the time step. In this case, the final value is 745 (Figure 48c). Because the end of the time step is between two of the times in the time-series, the interpolated value calculated for LINEAREND will be used in the calculation.

((0×1 day) + (0+1000) × (1 day)/2 + (1000 + 1150) × (3 days)/2) / (1 day + 1 day + 3 days) = 745.

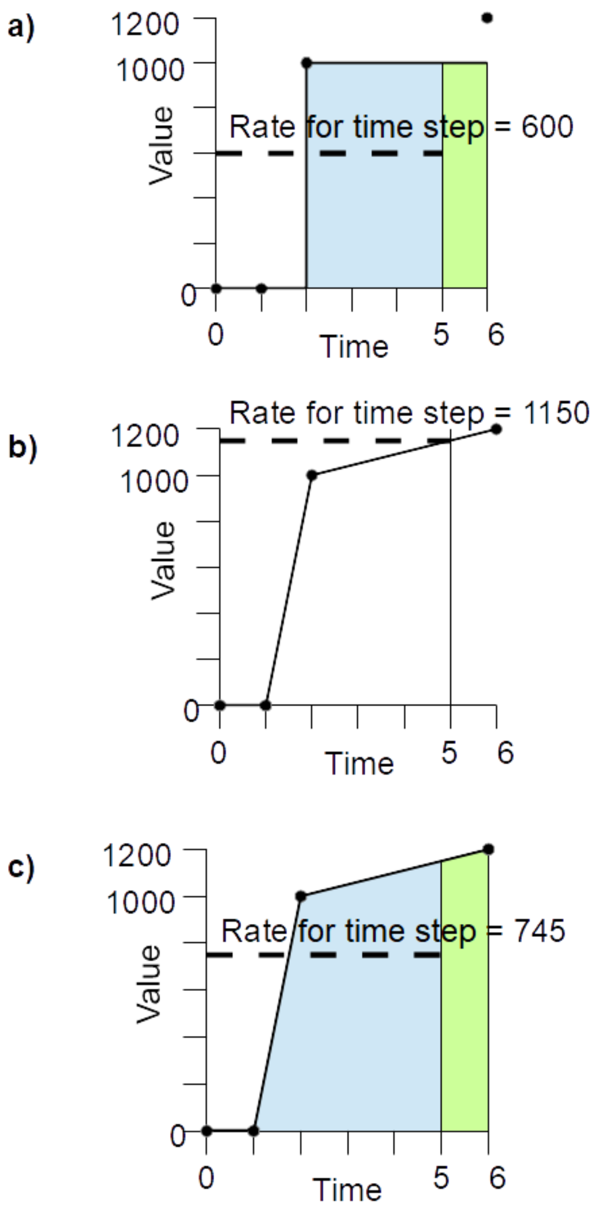


Figure 48 - Illustrations of interpolation methods for time series. The time step in the example extends from time = 0 to time = 5. The solid line represents the assumed change in the time series based on the interpolation method used. a) STEPWISE method - The value used for the time step is the average value for the time step assuming that the value changes abruptly at each time in the time series. b) LINEAREND method - The value used for the entire time step is determined by linear interpolation between times surrounding the time of the end of the time step rather than a time-averaged value. c) LINEAR method - The value used for the time step is the average value for the time step assuming that the value changes linearly between each time in the time series. For (a) and (c) a time-averaged weight is used and the blue area represents the area used in calculating the time-averaged weight. At time = 5, a new time step begins, shown by the green area.

The appropriate interpolation method to use will depend on how the value being modeled changes over time.

- STEPWISE could be useful for modeling pumping rates in the Well package where someone sets the pumping rate to a certain value and then leaves it at that value until there is a reason to change it. The STEPWISE method ensures that the volume of pumping in the time step is correct.
- LINEAREND would be appropriate for a specified head boundary condition in which the specified head is changing continuously. Because MODFLOW uses an implicit formulation, the specified head for a time step should be the head at the end of the time step.
- LINEAR might be appropriate for river stage where a gage records the river stage at a fixed interval and the stage is assumed to change continuously between the times at which the river stage was measured.

In addition to a name and an interpolation method, time series files can include scale factors for each time series. If scale factors are included, all the values in the time series are multiplied by the scale factor.

Time-series can be used with the following packages.

- CSUB: Skeletal Storage, Compaction, and Subsidence
- CHD: Constant Head Boundary
- WEL: Well
- DRN: Drain
- RIV: River
- GHB: General Head Boundary
- RCH: Recharge
- EVT: Evapotranspiration
- MAW: Multi-Aquifer Well
- SFR: Streamflow Routing
- LAK: Lake
- UZF: Unsaturated Zone Flow
- TVK: Time-Varying Hydraulic Conductivity
- TVS: Time-Varying Storage

10.2 Time-Array-Series Files

Time-array-series files are similar in concept to time-series files. However, each time-array-series associates an array of real-number values with each time rather than a single value. For example, in the Recharge package, an array of recharge rates can be used for each time step where the array specifies the recharge rate for every cell over the top surface of the model. These recharge rates can be interpolated from a time-array-series. Only a single series of arrays can be defined in one time-array-series file and each file can only be associated with one package. Each time-array series has an array, an interpolation method, and an optional scale factor.

Time-array-series can be used with the Recharge and Evapotranspiration packages.

11 Observation Utility

The Observation Utility is used to generate a time series of simulated values at various locations. Values are printed at the end of each time step. The user specifies an observation type of interest along with either one or two identifiers for each observation. For example, a cell could be an identifier for a head observation. MODFLOW saves the simulated value associated with the observation to a file at the end of every time step. There is no attempt made to interpolate heads, flows, or any other observation to a specific location within a cell or to a specific time within a time step. A separate program can be used to interpolate to the observation location and time. Mf6ObsExtractor.exe (distributed with ModelMuse) is one such program. There are over 100 different observation types that can be specified in MODFLOW 6. These include, head, drawdown, flow through a specific face of a cell, and flows related to boundary conditions such as drains, rivers, specified head cells, and general-head boundaries. The "Lake," "Streamflow Routing," "Multi-Aquifer Well," "Unsaturated Zone Flow," and "Skeletal Storage, Compaction, and Subsidence" packages each have multiple types of observation associated with them.

The user can specify whether the output from the Observation utility is to be a binary or text file. The binary files have the advantage of saving data without rounding errors that are introduced during the conversion of binary values to base-10 numbers. Output from the Observation utility is described within the file "mf6io.pdf" in the section titled "MODFLOW 6 - Description of Input and Output." The PDF is included in the doc folder distributed with MODFLOW 6 as described in Box 2 of this book.

12 Advanced Packages

MODFLOW has several packages that provide advanced simulation capabilities (Langevin et al., 2017; Hughes et al., 2022). These packages have complex input requirements which can make them difficult to use. In many cases, using them provides little benefit because a simpler representation of the features can work just as well with less effort. Only brief descriptions of their capabilities are provided here. In addition, there are several packages used for simulating solute transport (Langevin et al., 2022). The solute transport process in MODFLOW is outside the scope of this book so those packages are not described.

12.1 Skeletal Storage, Compaction, and Subsidence (CSUB) Package

The CSUB package simulates subsidence due to aquifer compaction. The compaction can be instantaneous or delayed and can be elastic or inelastic (Hughes et al., 2022). The compaction is assumed to occur in interbeds within a cell. The modeler needs to enter data that is related to compaction. Unless modeling subsidence is part of the goal of the project, the CSUB package is not required.

12.2 Multi-Aquifer Well (MAW) Package

The MAW package is designed for simulating wells that extend vertically over several cells although it can also be used for wells in a single cell (Langevin et al., 2017). The user must specify the extent of the well screen or screens, the conductance of the well screens, and the pumping rate among other inputs. If the pumping rate is zero, the MAW package can simulate flow between cells through the well bore. The MAW package allows MODFLOW to reduce the specified pumping rate in the well or shut off the well entirely if the head in the well gets too low. This might occur if the well pump cannot pump at the specified rate when the lift is too high.

12.3 Lake (LAK) Package

The Lake package is designed for simulating lakes whose water level is primarily controlled by groundwater flow into or out of the lake (Langevin et al., 2017). However, in many lakes, groundwater flow has little effect on the lake level. This is especially true of human-made lakes. Lakes in which lake levels are not greatly affected by groundwater flow can be simulated with other packages such as the General-Head Boundary or Constant Head packages. One way the Lake package might be used is to simulate water levels in an open-pit mine after closure.

12.4 Unsaturated Zone Flow (UZF) Package

The Unsaturated Zone Flow package simulates vertical flow in the unsaturated zone above the water table. The user specifies the infiltration at the ground surface and the UZF package simulates the flow of this water downward to the water table (Langevin et al., 2017). It can also simulate evapotranspiration. Beginners find this attractive as it seems like an easier way to simulate recharge than estimating the losses in the unsaturated zone themselves. However, there can be large uncertainties associated with the input required for the UZF package. These uncertainties can undermine the accuracy of the model. The UZF package can also greatly increase both the memory used by the model and the time required to run the simulation. Unless there is a long delay between when precipitation occurs and when the recharge reaches the saturated zone, the Recharge and Evapotranspiration packages are typically sufficient. Successful usage of the UZF package requires that the properties of the unsaturated zone are reasonably well known.

12.5 Water Mover (MVR) Package

The Water Mover package is used to simulate transfers of water from one boundary condition to another (Langevin et al., 2017). The user designates a source boundary and receiver boundary along with controls on the amount of water transferred. The source can be a boundary in one of the following packages:

- Well Package;
- Drain Package;
- River Package;
- General-Head Boundary Package;
- Multi-Aquifer Well Package;
- Streamflow Routing Package;
- Unsaturated Zone Flow Package; and
- Lake Package.

The receiver can be a boundary in one of the following packages.

- Multi-Aquifer Well Package;
- Streamflow Routing Package;
- Unsaturated Zone Flow Package; and
- Lake Package.

The Water Mover package can only be used if one of the other advanced packages is used.

12.6 Buoyancy (BUY6) Package

The Buoyancy package can be used to simulate the effects of fluid density differences on groundwater flow (Langevin et al., 2022). If it is used along with the groundwater transport process, the fluid density can be calculated from the solute concentration, but the fluid density can also be specified directly.

12.7 Viscosity VSC Package

The Viscosity package accounts for the effects of solute concentration on fluid viscosity and thus on hydraulic conductivity and the conductances in boundary conditions.

12.8 Time-Varying Hydraulic Conductivity (TVK) and Time-Varying Storage (TVS) Packages

The Time-varying Hydraulic Conductivity and Time-varying Storage packages allow the modeler to specify new values for any of the three hydraulic conductivity values or the storage parameters for a stress period. The new values remain in effect until changed. For example, if an earthquake caused changes in the aquifer properties in the fault zone, these changes could be simulated with the TVK and TVS packages.

12.9 Groundwater Flow (GWF) Exchange Package

MODFLOW allows the modeler to have separate grids in different parts of the model domain (Langevin et al., 2017). The Groundwater Flow Exchange package is used to simulate flow between the separate grids. It can also be used to simulate flow between widely separated parts of the same grid. MODFLOW allows but never requires use of separate grids.

13 Next Steps

This book presents basic information to help a groundwater professional start using MODFLOW 6. There is much more to know, and many resources are available. It is useful to read additional material to learn more about using MODFLOW 6. For example, there are two public-domain publications from the USGS that discuss modeling concepts applicable not only to MODFLOW but to modeling in general. The first discusses conceptualization of groundwater systems and their boundaries (Reilly, 2001↗) and the other presents guidelines for evaluating ground water flow models (Reilly and Harbaugh, 2004↗). Reading the guidelines for evaluating models helps a modeler develop better models using MODFLOW.

It is also useful to read other MODFLOW model reports. The USGS has published reports that use MODFLOW and many are listed on the Community Surface Dynamics Modeling System's MODFLOW references web page↗. New model reports and data sets from the USGS are listed in the USGS Groundwater Highlights newsletter. One can subscribe to the USGS newsletter at https://water.usgs.gov/ogw/highlights/↗. In addition, the Google group devoted to MODFLOW users at https://groups.google.com/g/modflow↗ may be of interest to some readers.

Saul Montoya's blog on https://hatarilabs.com/ih-en↗ has many tutorials about MODFLOW. The Hatari Labs' YouTube channel includes numerous videos about MODFLOW. Experimenting with MODFLOW software even on hypothetical systems is a good way to master the program. Following guidance from other resources that offer example MODFLOW modeling problems is an excellent way to learn more. Another Groundwater Project book, Groundwater Resource Development: Effects and Sustainability↗ provides exercises that use MODFLOW to assess the effects of groundwater resource development on baseflow in streams. Viewing and following along with videos at https://www.usgs.gov/mission-areas/water-resources/modelmuse-tutorial-videos-0↗ is another way to advance one's knowledge of MODFLOW.

Finally, there is one more exercise included in this book, Exercise 8↴. Unlike, the previous exercises, it represents a more realistic situation and includes some real data.

14 Exercises

Exercise 1

Open the Name file (e.g., SimpleModel.nam) for the model you created and named based on the instructions of Section 4. It can be opened with any text editor (e.g., Notepad). In the Name file, identify the discretization file for the model. It will be listed in the Packages section with the identifier "DIS6." Open the discretization file with a text editor. Also open the file mf6io.pdf. Mf6io.pdf will be in the "doc" directory of the MODFLOW 6 distribution. In mf6io.pdf locate the description of the Structured Discretization (DIS) input file. It can be found in the section named "Groundwater Flow (GWF) Model Input." Go to the section titled "Explanation of Variables" and note the blocks in your file and the description of the variables as listed in mf6io.pdf. Any line that begins with "#", "!", or "//" indicates it is a comment and does not affect the meaning of the file.

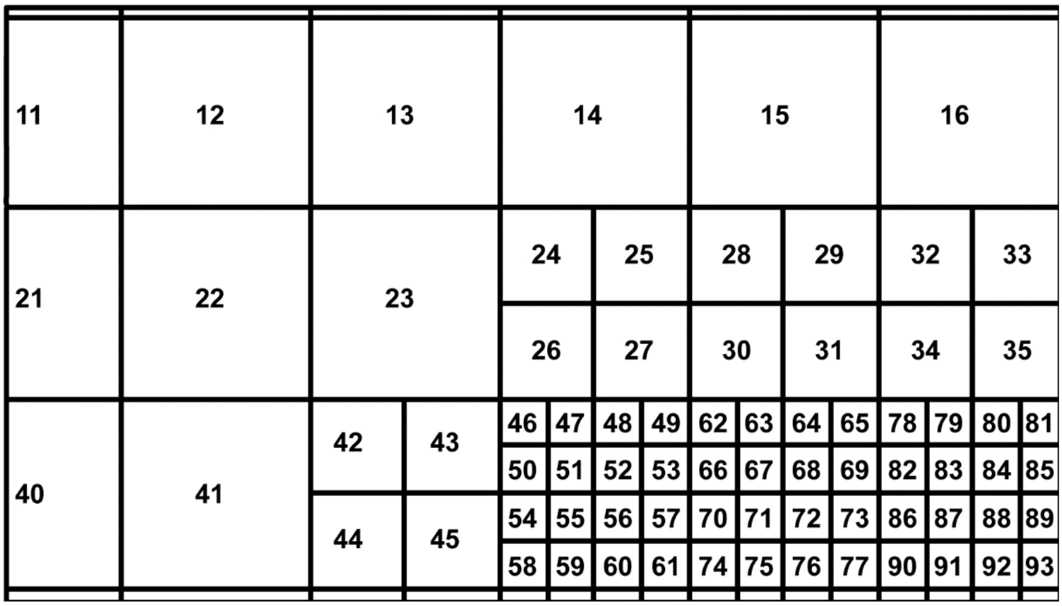
MODFLOW has an option to save a binary grid file that can be used by other programs. In the section of mf6io.pdf that describes the structured grid file, determine how this option is specified. Then examine your structured grid file and determine whether a binary grid file will be saved for your model.

Solution to Exercise 1↴

Return to where text links to Exercise 1↥

Exercise 2

This image shows part of a MODFLOW model that uses Discretization with Vertices. Each cell can have flow between it and its neighbors.



Part of a DISV grid with cell numbers.

a. What cells can have flow from or to cell 23?
b. What cells can have flow from or to cell 25?
c. What cells can have flow from or to cell 46?

Solution to Exercise 2↴

Return to where text links to Exercise 2↑

Exercise 3

Introduction to the Basic, Node Property Flow, and Constant Head Packages

Exercises 3 through 7 were originally developed for courses on MODFLOW taught within the USGS by Arlen Harbaugh, David Pollock and others starting in 1982. The exercises are adapted here for use with MODFLOW 6.

This groundwater flow system is in a valley which is 10,000 m wide and many kilometers long. An unconfined upper sand aquifer is separated from a lower sand aquifer by a fine-grained silt that acts as a confining layer for the lower aquifer. The aquifers are bounded on the east and the west by impermeable hills. The lower aquifer is underlain by impermeable bedrock. Along the eastern boundary there is a river that fully penetrates the upper aquifer, with a stage of essentially 320 m along the entire distance from the northern to the southern moraine. The northern boundary of the flow system is a low permeability moraine 20 km upstream. The southern boundary is also a low permeability moraine located 20 km downstream. On the western boundary there is a canal that fully penetrates the upper aquifer, with a stage of essentially 330 m along the entire distance from the northern to the southern moraine. The canal receives its water from the river north of the northern moraine and discharges to the river south of the southern moraine.

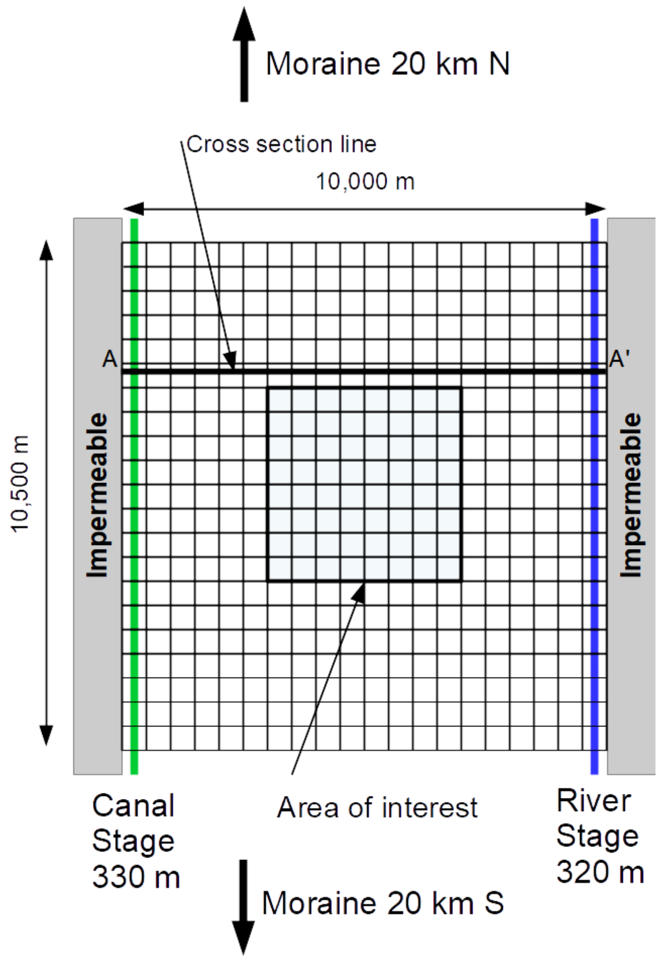


Illustration of model configuration in map view.

In the upper aquifer, the horizontal hydraulic conductivity is isotropic with a value of 50 m/d, the vertical hydraulic conductivity is 10 m/d and the specific yield is 0.20. The bottom of the upper aquifer is at an elevation of 220 m. The top is at land surface, which can be specified as 350 m. (The river and canal are treated as fully penetrating the upper aquifer to illustrate how to set up the model. Such deep waterways are not typical of the real world.)

In the silt unit, both the horizontal and vertical conductivity are 0.01 m/d and the specific storage is 0.000001 m^{-1}. The bottom of the silt layer is at 200 m.

In the lower aquifer the horizontal conductivity is isotropic with a value of 200 m/d, the vertical conductivity is 20 m/d and the specific storage is 0.000001 m^{-1}. The bottom of the lower aquifer is at 0 m.

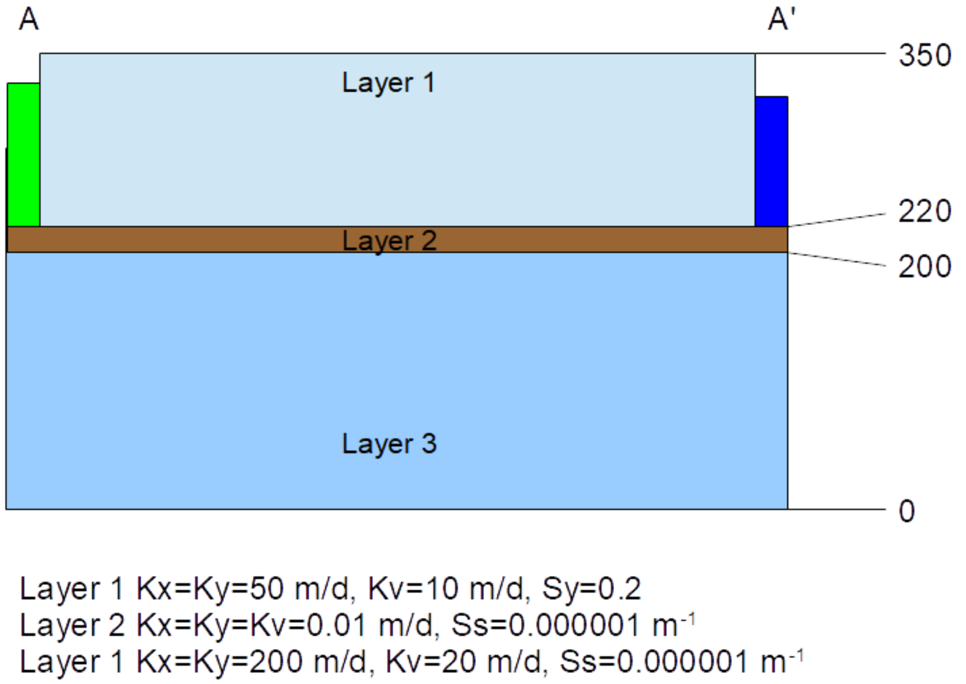


Illustration of cross section view.

The area of interest is a square farm 4,000 m on a side which is halfway between the canal and the river. The modeled area is larger than the area of interest so as to include natural boundaries of the system. The river and canal form natural boundaries. As formulated here, the northern and southern boundaries (see map view above) will be along flow lines, thus no flow will cross these boundaries. In later versions of this model additional stresses such as wells will be introduced. The aim is to place the northern and southern boundaries far enough away from the area of interest that the results in the area of interest will not be greatly affected.

Simulate groundwater flow in the vicinity of the field as a steady-state flow system. Use 3 layers, 21 rows and 20 columns with a uniform horizontal grid spacing of 500 m in each direction. Represent the upper aquifer with layer 1, the silt with layer 2, and the lower aquifer with layer 3. The northern and southern boundaries will be presumed streamlines perpendicular to the canal and river. The eastern and western boundaries will be constant head boundaries representing the river and the canal. The head in the river is 320 m the head in the canal is 330 m.

Return to where text links to Exercise 3↥

Creating the Base Run for Exercise 3

The following numbered items provide general directions for creating the base model. Some readers may want to undertake the work using only those items. For readers who prefer more detailed guidance, step-by-step instructions for creating the base model are provided in Box 5↧.

1. Create a model that has 3 layers, 21 rows, and 20 columns. Save the model in a directory you create for this exercise. Name the model PS1A.
2. Enter all the data relating to the description of the flow system using the DIS, NPF, STO, and TDIS packages. Optional: decrease the nonlinear and linear head closure criterion in the IMS package to 0.0001 and 1×10^{-5} respectively. Including data for the Storage package is optional for this model because it is a steady-state model. However, later exercises involve versions of this model that will require storage properties, so it is included here.
3. Define specified head boundaries in the upper aquifer (layer 1) on the east and west edges of the model to represent the canal and the river.
4. In the Output Control make the following changes:
    a. Change the Head Print Format Code to 5 (FORTRAN format 9G13.6). This format causes output to be printed with 9 values per line, 13 characters per value including up to 6 decimal digits for each value.
    b. Change the option to save heads so that the heads are saved at the end of the simulation. This setting will cause MODFLOW to generate a binary file head output file that can be read by post-processing programs such as graphics applications.
5. Run the model with MODFLOW 6.
6. Examine the text output of results by opening output file named PS1A.LST in a text editor.
7. If using ModelMuse to work with the model, examine the visual output of calculated heads by going to *File|Import|Model Results…* and selecting the binary output file named PS1A.bhd that contains the calculated heads and choose the contour grid with the head values. Colored coded and labeled values of head will appear on the grid. The plan-view contours reflect horizontal flow from the canal to the river, and the cross-sectional view at the bottom of the screen indicates downward flow from the canal to layer 2 and subsequently layer 3 as well as upward flow to the river.

8. Examine the visual output of calculated flows by going to *File|Import|Model Results…* and selecting the binary output file named PS1A.cbc that contains the calculated flows and choose to color the grid with the flow values for the specified head (CHD) boundaries. Inflow from the canal is colored in red and outflow to the river is colored in blue. When the cursor is over one of the colored cells, the flow rate through the specified head cell is displayed on the far-right panel of the status bar. More information about the cell under the cursor is provided in the *Data|Show Grid or Mesh Values* dialog box.

After completing the simulation, use the output to answer the questions posed in exercises 3.1 through 3.4.

Exercise 3.1

Using the water budget summary at the end of the MODFLOW listing file, what is the total volumetric rate of flow leaving the canal and the total volumetric rate of flow entering the river? Round to the nearest whole number.

For comparison, use Darcy's Law and the head output in the listing file to compute the total flow rate out of the canal and into the river. Calculate the flow from one canal cell and multiply by 21 rows to get the total flow. To simplify the calculation, use the average saturated thickness between columns 1 and 2 to compute the horizontal flow. For the vertical flow calculation, assume that all the head difference between the canal and the node in layer 2 occurs within the confining layer between the bottom of layer 1 and the node in layer 2. How do these flow rates compare with those determined from the MODFLOW water budget summary?

Solution to Exercise 3.1⤵

Exercise 3.2

Between which two columns does the flow between layer 1 and layer 2 change from downward to upward? What about between layers 2 and 3?

Solution to Exercise 3.2⤵

Exercise 3.3

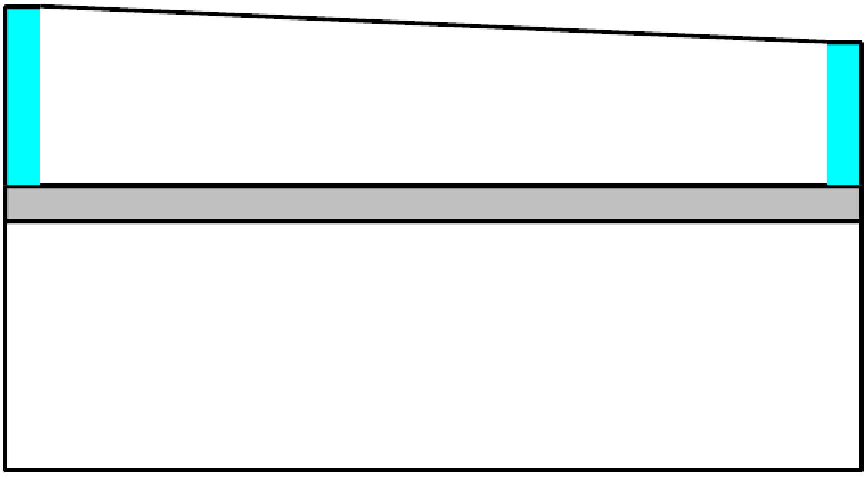
What are the total volumetric rates of flow from column 10 to column 11 in each of the three model layers? Use Darcy's Law and the head output from the MODFLOW listing file to make these calculations and fill in the table provided here.

	h_{10}	h_{11}	Δh (h_{11} - h_{10})	b Layer thickness	Q (All 21 rows)
Layer 1					
Layer 2					
Layer 3					

Solution to Exercise 3.3⤵

Exercise 3.4

What is the total volumetric rate of flow downward across the top of layer 3 for columns 1 through 10? What is the total volumetric rate for flow upward across the top of layer 3 for columns 11 through 20? *Hint*: Use a simple water balance equation and the information from your answers to the previous exercises. Show your results on the schematic cross section provided here.



Explain how you could use Darcy's law as an alternative approach for calculating the total downward and upward flow rates described above. You do not need to make the calculations; just describe how you would do it.

Solution to Exercise 3.4⤵

Exercise 3.5

In some cases, flow rates through part of the model may be of interest but calculating them through a simple water balance approach might not be practical. ZONEBUDGET can be used to address such questions. With ZONEBUDGET, the modeler assigns cells to individual zones and ZONEBUDGET computes a water budget for each of those zones. ZONEBUDGET is distributed with MODFLOW 6. For this exercise, use ZONEBUDGET to calculate the answer to the same question as before; determine the flow downward from layers 2 to 3 in columns 1 through 10 and the upward flow in columns 11 through 20 from layers 3 to 2. Step-by-step instructions for PS1B1 are provided in Box 6⤵.

Solution to Exercise 3.5⤵

Return to where text links to Exercise 3⤴

Exercise 4

Exercises 4.1 through 4.4 each involve a modification of the base case model, PS1A from Exercise 3. In each case, use the base run model (PS1A) as a starting point, make the changes, then save a separate modified model with the new model name as indicated for each exercise (PS1B1, PS1B2, PS1B3, and PS1B4).

Return to where text links to Exercise 4↥

Exercise 4.1

Start by opening model PS1A and saving it as PS1B1. Next double all the hydraulic conductivity values (vertical and horizontal) in both aquifers and the confining layer, then save and run the simulation again. Step-by-step instructions for PS1B1 are provided in Box 7↴.

How much water is leaving the canal? What is the effect of increasing the hydraulic conductivities on the head distribution?

Solution to Exercise 4.1↴

Exercise 4.2

Start by opening model PS1A and saving it as PS1B2. Next, raise the heads in the canal and in the river by 1 m and run the simulation again using the hydraulic conductivities used in the original problem described in Exercise 3 Step-by-step instructions for PS1B2 are provided in Box 8↴.

What is the effect on the head distribution? How much water is leaving the canal? Calculate the rate of flow between column 10 and column 11 for layer 1 and layer 3.

Solution to Exercise 4.2↴

Exercise 4.3

Start by opening model PS1A and saving it as PS1B3. Next, reduce the vertical and horizontal hydraulic conductivity of the silt to 0.0001 m/d, then save and run the model. Step-by-step instructions are provided in Box 9↴.

What is the effect on the heads and flow rates in each layer? Calculate the rate of flow between column 10 and column 11 for layer 1 and layer 3.

Solution to Exercise 4.3↴

Exercise 4.4

Start by opening model PS1A and saving it as PS1B4. Next, add a high-conductivity region to layer 1 by setting the hydraulic conductivity equal to 250 m/d for a rectangular block of cells extending from (row 9, column 5) to (row 13, column 15), then save and run the model. Step-by-step instructions for PS1B4 are provided in Box 10↴.

Observe the change to the head distribution in layer 1. As a reminder, if you are using ModelMuse, the heads can be displayed by selecting *File*|*Import*|*Model Results* and selecting the binary head output file, then choosing to color the grid or to contour the heads.

Solution to Exercise 4.4↴

Return to where text links to Exercise 4⮥

Exercise 5

Introduction to Stress Packages

This exercise introduces the use of stress package features. The groundwater flow system is like the base case presented in Exercise 3 (PS1A), except the canal on the left side has been removed and areal recharge has been added to layer 1 as the source of water for the system. The recharge rate is 0.005 m/day.

The exercise consists of four major parts (A, B, C, and D). Part A introduces the use of the recharge package. Parts B, C, and D demonstrate the river, drain, general-head boundary, and well packages.

Return where text links to Exercise 5↥

Part A - The Recharge Package

1. Open model PS1A and use the Save As option to create a new model named PS2A.
2. Make the changes required to remove the canal and add recharge.
3. Run MODFLOW, then view the resulting heads in ModelMuse and the listed output in the ".lst" file and use the results to complete the exercises 5.1 through 5.9.
4. Step-by-step instructions are provided in Box 11↧.

Exercise 5.1

Does the recharge in the budget printed in the listing file match the recharge the amount of recharge applied? Hint: Multiply the recharge rate by the model area.

Solution to Exercise 5.1↧

Exercise 5.2

How much water is going into the river?

Solution to Exercise 5.2↧

Exercise 5.3

How is the head distribution in this simulation different from that in problem set 1? What causes the difference?

Solution to Exercise 5.3↧

Exercise 5.4

Between which two columns does the flow change from downward across the bottom of layer 1 to upward across the bottom of layer 1?

Solution to Exercise 5.4↧

Exercise 5.5

For each of the three model layers, calculate the total volumetric rates of flow between the two columns identified in Exercise 5.4. The provided table is useful for compiling the information.

	h_{left}	h_{right}	Δh $h_{right} - h_{left}$	b Thickness	Q (All 21 rows)
Layer 1					
Layer 2					
Layer 3					

Solution to Exercise 5.5

Exercise 5.6

Calculate a budget for cell 1,1,10 (layer, row, column) and for cell 1,1,20, by calculating all flows into and out of each cell and the mass balance error.

Solution to Exercise 5.6

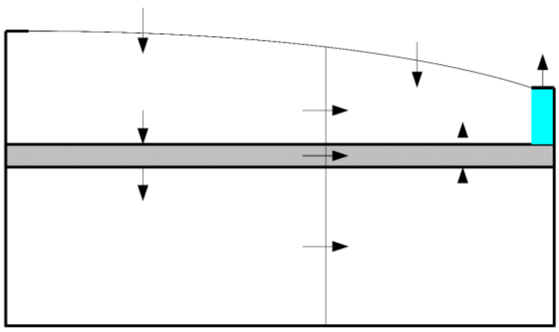
Exercise 5.7

Double all the vertical and horizontal hydraulic conductivity terms in all layers. In ModelMuse, this is done using *Data|Edit Data Sets*. Name this model PS2A1. Describe the changes in the flow system.

Solution to Exercise 5.7

Exercise 5.8

Using the results from PS2A (the original run with the recharge package), calculate a water budget, for each of the sub-regions indicated on the cross section below. Label the flow arrows on the cross section with the values from the simulation.



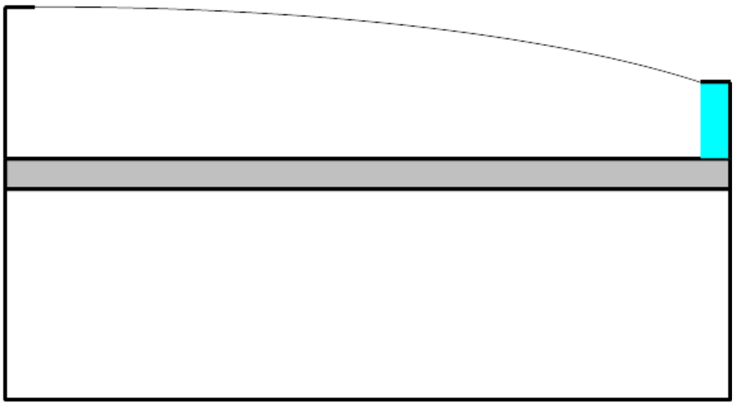
Diagrammatic model cross section with Recharge package.

Solution to Exercise 5.8

Exercise 5.9

All the water that enters the system comes from recharge entering the top of the system. Delineate the zone that recharges model layer 3 on the schematic cross section shown in image below.

Hint: Flow in the system is from left to right. The recharge source area for layer 3 starts at the west edge of the system. Using water budget information from previous Exercise 5 solutions, calculate the width of the source area required to capture enough recharge to supply the flow that enters layer 3.



Diagrammatic unconfined model cross section.

Solution to Exercise 5.9⤵

Part B - The River Package

This exercise creates the same model as PS2A except that the constant-head river on the east side of the grid is represented using the MODFLOW river package. The hydraulic conductivity of the riverbed is specified as 20 m/d, and the width is 10 m. The elevation of the riverbed bottom is 317 m. The riverbed is 1 m thick. Create model PS2B as follows.

1. Open model PS2A in your GUI and use the Save As option to create a new model named PS2B.
2. Make the changes are required to delete the CHD representation of the river and replace it with the River Package.
3. Run MODFLOW then view the resulting heads in ModelMuse and the listed output in the ".lst" file.
4. Step-by-step instructions are provided in Step-byStep Instructions for Model PS1B in Box 12⤵.

When the model simulation is successful, then it be used for Exercise 5.10.

Exercise 5.10

Calculate the cell budget for cell 1,1,20 (layer, row, column). Compare to the budget for the same cell in run PS2A which was completed in Exercise 5.6. How does the use of the river package represent a different conceptual model?

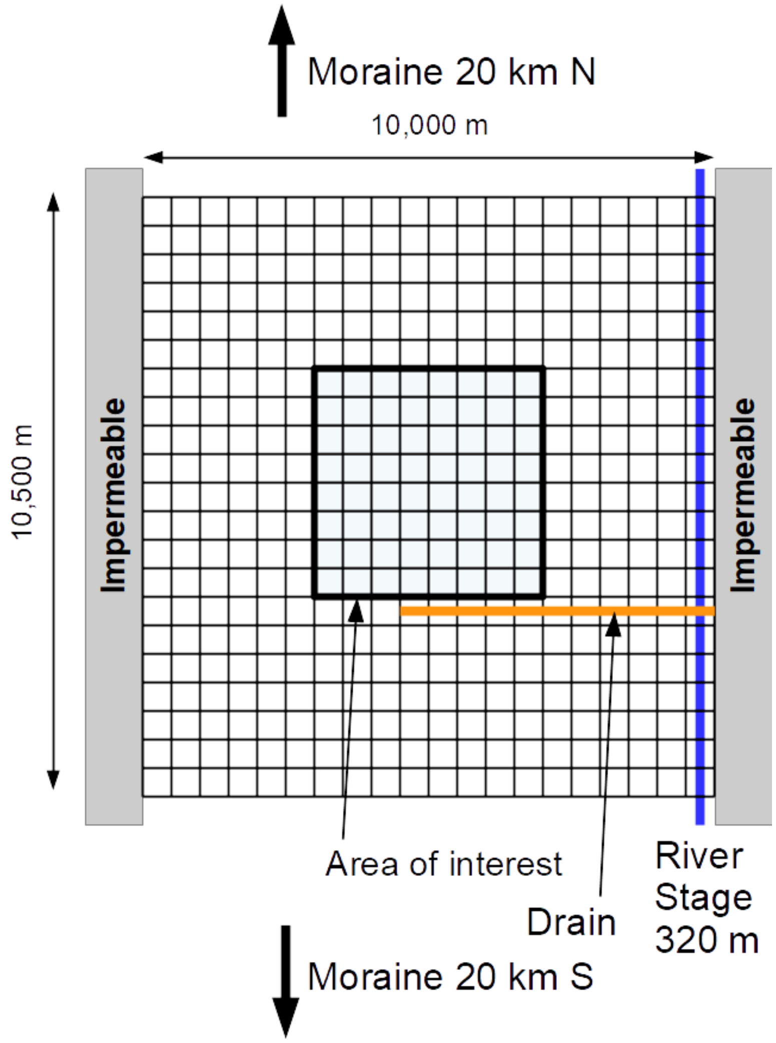
Solution to Exercise 5.10⤵

Part C - The Drain and General-Head Boundary Packages

This exercise uses the same system as model PS2B except that a buried drain tile, simulated with the DRAIN package, is installed in row 15 in columns 10-20 (coordinates = (4500, -7250) to (10,000, -7250)) as shown in the image below. The conductance between the aquifer and the drain is 100,000 m^2/d, the elevation of the drain is 322.5 m.

Follow these steps:

1. Open model PS2B in your GUI and use the Save As option to create a new model named PS2C.
2. Make whatever changes are required to add the drain.
3. Run MODFLOW then view the resulting heads in ModelMuse and the listed output in the ".lst" file.
4. Examine the listing file to notice the impact that the drain has on water levels.
5. Step-by-step instructions are provided in Box 13↴.



ation of drain boundary.

When you have a successful run, it can be used for Exercises 5.11 and 5.12.

Exercise 5.11

The presence of the drain cells will affect the groundwater levels. If the discharge out of the drain cells is measured, that observation can be used to help estimate model parameters such as the drain conductance. For this exercise, construct a cell budget for cells (1, 15, 10) and (1, 15, 19). You may also wish to plot the heads in a graphical user interface to see the overall effect of the drains.

Solution to Exercise 5.11↴

Exercise 5.12

Replace the drain with a general head boundary using the same head and conductance. Name this model PS2C1.

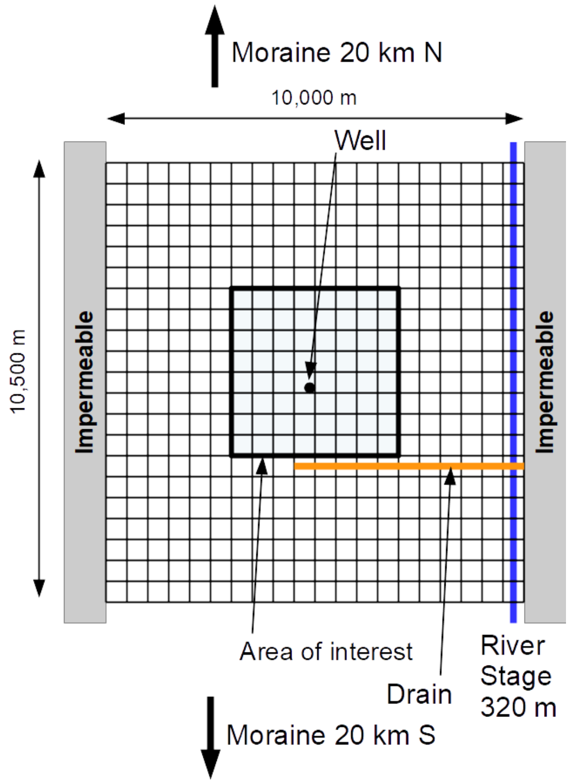
Step-by-step instructions are provided in Box 14↴.

Compare the water budget for run PS2C1 with the water budget for run PS2C and explain any difference you observe.

Solution to Exercise 5.12↴

Part D - The Well Package

This exercise uses model PS2C with the addition of a well located in layer 1 at coordinates 4875,-5375 in the southeastern quarter of the cell at row 11, column 10 as shown below. The discharge rate of the well is 75,000 m^3/d. Steps for the exercise follow.



Location of added well.

1. Open model PS2C in your GUI and use the Save As option to create a new model named PS2D.
2. Make the model changes required to add the well.
3. Run MODFLOW then view the resulting heads in your GUI. If you contour the heads notice that the contours are not centered on the well itself but on the center of the cell containing the well. This is because the heads are only calculated at the centers of cells. If more detail is needed, the cells need to be made smaller.

4. Examine the listing file to see the values of head in nodes around the well and the water budget.
5. Step-by-step instructions are provided in Box 15↴.

When you have a successful run, it can be used to complete Exercise 5.13, and a successful run of Exercise 5.13 can be used for Exercise 5.14. A common error is specifying the well withdrawal as a positive value. This is often discovered by noticing that heads increased after adding a well so, it is always good practice to view results and confirm they are consistent with your concept of how the model should behave.

Exercise 5.13

What is the change in the total volumetric rate of flow to the river in run PS2D compared with that of run PS2C? What is the change in the total volumetric rate of flow to the drain in run PS2D compared with that in run PS2C? Add the change in river discharge to the change in drain discharge and compare the sum to the well discharge rate. Record your results in the table provided. Round the values to the nearest whole number.

Flow rate out	PS2D	PS2C	PS2D - PS2C
River			
Drain			
Well	75,000	0	75,000
			Sum = 0

How much is the water table drawdown after the pumping reaches steady state?

Solution to Exercise 5.13↴

Exercise 5.14

Starting with model PS2D, use the Save As option in your GUI to create a new model named PS2D1. Add a second pumping well in layer 3, at coordinates (2374, -6237) that has a withdrawal rate of 100,000 m^3/d.

Step-by-step instructions are provided in Box 16↴.

How do the head distribution and flow budget change between models PS2D and PS2D1? How much drawdown is caused by the second well?

Use a contouring program or your GUI to display the head distribution in layer 1 and layer 3. Draw a sketch of what you think the source area at the water table looks like for the well in layer 3.

Solution to Exercise 5.14↴

Return to where text links to Exercise 5↥

Exercise 6

Introduction to Effects of Model Boundaries and Simulation of Transient Flow

This exercise examines the effect of boundary conditions on groundwater flow using a system like the one represented by model PS2C. Recall that in model PS2C, the source of water is uniform areal recharge and discharge occurs at a river along the east side of the system. Because groundwater flow is parallel to the north and south boundaries, those boundaries are streamlines that can be simulated as no-flow boundaries in the model. Previously, when we added a pumping well near the center of the system in the upper aquifer, we continued to represent the north and south boundaries as no-flow boundaries even though we know those boundaries would not continue to act as perfect no-flow boundaries in nature. Head gradients would change, and some water would flow across those boundaries. How much flow would be induced across the north and south boundaries? What error is introduced by treating those boundaries as no-flow? What other choices are available to represent the north and south boundaries? And, how do the different boundary condition options compare to one another? Questions such as these arise in nearly every model study. This simple problem provides an opportunity to examine the impact of boundary conditions in detail. It also serves as an introduction to transient flow simulation using MODFLOW.

Although the system simulated in this exercise is like that of model PS2C, it has been simplified as a one-layer model in order to make it easier to analyze the budget components of the transient system as well as the impact of boundary conditions. The model properties are as follows.

- 1 layer (Convertible, bottom elevation = 0 m)
- 21 rows and 20 columns
- Uniform 500 m areal grid spacing
- Horizontal hydraulic conductivity = 50 m/d
- Vertical hydraulic conductivity = 10 m/d (this value must be entered in MODFLOW to fully define the material properties, but is irrelevant in a one-layer model because there is no component of flow in the vertical direction)
- Specific yield = 0.2
- Specific storage = 0.000001 m^{-1}
- Uniform areal recharge of 0.0025 m/d
- Discharge to a river located at the right edge of the model (X = 9800) in column 20, rows 1-21
- River stage = 50 m
- River bed bottom = 48 m
- River bed conductance = 100,000 m^2/d
- A well located at (4820, ⁻5297) in row 11, column 10 discharging at 100,000 m^3/d

This exercise consists of three model simulations.

1. Base simulation – steady-state flow with no well discharge.
2. Simulation A – transient flow with a discharge well and constant head boundaries on the north and south. The specified assigned values equal to the heads calculated for the base simulation.
3. Simulation B – transient flow with discharge well and no-flow boundaries on the north and south.

The base simulation is necessary to establish the starting heads for the transient simulations A and B. The head, drawdown, and water budget output from simulations A and B will be examined and compared to answer several questions about the effects of the boundary conditions on the system.

Return to where text links to Exercise 6↥

Creating the Base Simulation - Steady-State Flow with No Well Discharge

Steady-state heads are needed as starting heads for the constant-head cells along the north and south boundaries in run A. These steady-state heads also provide meaningful starting heads for the transient flow simulations of runs A and B. This simulation will save the steady-state heads calculated by the base simulation in an unformatted binary file. The binary file cannot be edited or displayed on the screen with a text editor, but it can be processed by MODFLOW to provide starting heads for simulations A and B.

Rather than build this model (PS3Base), the PS3Base files can be extracted from the zip file provided on the book page Getting-Started-With-MODFLOW_Exercises.zip↗, then imported to a Graphical User Interface. Examine the contents of the model folder, then run MODFLOW to generate its output, import the calculated heads to visualize the head distribution, and view the results in the listing file. The model PS3Base is supplied both in the form of a ModelMuse file and as the MODFLOW input files. If you are working through the exercises using ModelMuse, you can start with the ModelMuse file. Otherwise, use the model input files.

Step-by-step instructions for starting the PS3Base model are provided in Box 17↴.

Using the heads calculated by the base model, two simulations will be conducted, one using constant head boundaries along the north and south border and the other using no-flow boundaries. Both start by calculating steady state heads without pumping, then simulate pumping for 1800 days, and finally solve for the steady condition with pumping which helps the modeler determine if the system was close to steady state after 1800 days of pumping.

Simulation A – Constant Head Boundaries

The following steps outline construction of the model with constant head boundaries along the north and south boundaries. More detailed description of the steps is provided in Box 18↴.

1. Open model PS3BASE in a GUI and use the Save As option to create a new model named PS3A.
2. In the TDIS package, define three stress periods with the following properties. The MODFLOW storage package will be activated because the model has a transient stress period.
3. Stress period 1: length = 1 day, time steps = 1, multiplier = 1, type = SS
4. Stress period 2: length = 1800 days (about 5 years), maximum first time step size of 16 days which will result in 10 time steps given a multiplier of 1.5, type = TR
5. Stress period 3: length = 1 day, time steps = 1, multiplier = 1, type = SS
6. Make the changes required to specify constant heads along the north and south boundaries (row 1 and row 21) as described above.
7. Change starting head to be read from file PS3BASE.bhd. The format must be (BINARY).
8. For stress periods 2 and 3, add a well to row 11, column 10 with a discharge rate of -100,000 m^3/d. The well is not present in stress period 1.
9. Make the necessary modifications in the RIVER and RECHARGE packages to set all their properties to the same values for all three stress periods.
10. Modify the Output Control to make the necessary changes to save and print head at all time steps. By saving head in an output file we will be able to use graphics programs to display the output.
11. Step-by-step instructions are provided for PS3A in Box 18↴.
12. Run the model. ModelMonitor now shows the mass balance for 12 times steps. If a box displaying warnings appears, review the items listed and make corrections to the input if there are errors. There will be a warning message that there is no well boundary condition in the first stress period, but that is not a concern because we do not intend for the well to be pumping during the intial steady state period.
13. Import the results for all time steps to visualize the resulting head changes over time.

- *Stress Period 1*: The first stress period is a steady-state period with one time-step. Heads for this time step should be identical to those from PS3BASE because the hydraulic properties and stresses that define the ground-water flow system during this steady-state time step are identical to those in simulation PS3BASE. If that is true, why even include this first stress period? The answer has to do with the effect on the water budget when rows 1 and 21 are changed to constant head cells. Recall that areal recharge is not applied to constant head cells. That means that changing two rows in the model to constant head has the effect of decreasing the total recharge to the system. By including this first steady-state time step, the base simulation is reproduced for rows 2 through 20 but MODFLOW computes the amount of recharge for a smaller active flow area of the model (rows 2-20, columns 1-20) than in model PS3BASE. This step could have been skipped, and initial heads for the first transient stress period specified to be the head output from PS3BASE, but it was included because it makes it easier to examine how the water budget components change in response to the addition of the well discharge. It also makes it easier to compare the results of this simulation with those that will be generated later in simulation B, which treats rows 1 and 21 as no-flow cells rather than constant head cells.

- *Stress Period 2*: The second stress period is an 1800-day transient period represented by 10 time-steps. In this period, a discharge well withdraws water at a rate of 100,000 m^3/d. The heads in these 10 time-steps show the transient response of the ground-water flow system to the instantaneous addition of the discharge well. A modeler should have expectations for the nature of the results so that their expectations can be compared to model results and if they are not consistent the modeler can determine whether the model input needs to be corrected or their expectations were not reasonable. The logical expectations based on hydrogeologic knowledge would be that at early time, the volume budget will show a large rate of flow in from storage (because initially all the well discharge comes from storage), and little change in the rates of flow in from the constant head boundaries and flow out to the river (because it will take a while for heads near the boundaries to change). As time goes on, the rate of flow in from storage is expected to decrease (because more and more of the well discharge is drawn inward from the model boundaries) so the rate of flow in from the constant head boundaries is expected to increase and the rate of flow out to the river is expected to decrease.
- *Stress Period 3*: The third stress period is another steady-state period with one time-step with the discharge well pumping at the same rate as in period 2 (100,000 m^3/d). Later, examination of the output from the transient stress period will reveal that the system does not reach a new steady state at the end of the second stress period. Adding the final steady-state stress period provides a way to calculate the new steady-state head distribution in a single time step without the need for several additional transient time steps.

Exercise 6.1

Examine the output in the listing file, PS3A.LST and compare the water-budgets for the steady-state condition without the well (period 1, step 1) with the steady-state condition with the well (period 3, step 1). Record your results in the table provided here.

Model PS3A	**Flow rate (round to the nearest whole number)**		
Budget component	Steady-state, no well (period 1, step 1)	Steady-state with well (period 3, step 1)	Change in flow rate (with well – without well)
Inflow			
Storage-specific storage			
Storage-specific yield			
Constant head			
Wells			
River leakage			
Recharge			
Total			
Outflow			
Storage-specific storage			
Storage-specific yield			
Constant head			
Wells			
River leakage			
Recharge			
Total			
Inflow – outflow			

Solution to Exercise 6.1↴

Exercise 6.2

Compute the change in each budget component of simulation PS3A at time steps 1, 3, 5, 8, and 10 in stress period 2. You may find it convenient to fill out the flow rates themselves in the first table. Record your changes in flow rates in the second table provided here.

Model PS3A	Flow rate					
Budget component	Period 1, step 1	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow						
Storage-specific storage						
Storage-specific yield						
Constant head						
Wells						
River leakage						
Recharge						
Total inflow						
Outflow						
Storage-specific storage						
Storage-specific yield						
Constant head						
Wells						
River leakage						
Recharge						
Total outflow						
Inflow – outflow						

Model PS3A	Change in flow rate[1]				
Budget component	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow					
Storage-specific storage					
Storage-specific yield					
Constant head					
Wells					
River leakage					
Recharge					
Total inflow change					
Outflow					
Storage-specific storage					
Storage-specific yield					
Constant head					
Wells					
River leakage					
Recharge					
Total outflow change					
Inflow – outflow					

[1] Rate at the current time step minus the initial rate at period 1, step 1. Round to the nearest whole number.

Consider whether the volume budget summaries for various times (e.g., the end of the first steady state period, midway through the transient pumping period, at the end of the pumping period, and at the end of the steady state pumping period) are consistent with your expectations for how the system would behave.

Solution to Exercise 6.2⤵

Simulation B – No-Flow Boundaries

In this exercise the constant head cells of simulation PS3A are changed to no-flow cells to simulate the north and south boundaries as flow lines and keep the active flow area of the model the same as in PS3A. Start with model PS3A and use the Save As option to create a new model named PS3B. Change the specified heads for rows 1 and 21 to inactive cells to flag them as no-flow cells. Then run the simulation.

Step-by-step instructions for creating simulation PS3B are provided in Box 19⤵.

Exercise 6.3

Compare the water-budgets for the steady-state condition without the well (period 1, step 1) with the steady-state condition with the well (period 3, step 1). Record your results in the table provided here. How do they differ from PS3A?

Model PS3B	**Flow rate (round to the nearest whole number)**		
Budget component	Steady-state, No well (period 1, step 1)	Steady-state with well {Period 3, step 1)	Change in flow rate (with well - without well)
	Inflow		
Storage-specific storage			
Storage-specific yield			
Wells			
River leakage			
Recharge			
Total			
	Outflow		
Storage-specific storage			
Storage-specific yield			
Wells			
River leakage			
Recharge			
Total			
Inflow – outflow			

Solution to Exercise 6.3⤵

Exercise 6.4

Compute the change in each budget component of run PS3B at time steps 1, 3, 5, 8, and 10 in stress period 2. Record your results in the table provided here. You may find it convenient to fill out the flow rates themselves in the first table. Record your changes in flow rates in the second table provided here. How do they differ from PS3A?

Model PS3B	**Flow rate**					
Budget component	Period 1, step 1	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow						
Storage-specific storage						
Storage-specific yield						
Wells						
River leakage						
Recharge						
Total inflow						
Outflow						
Storage-specific storage						
Storage-specific yield						
Wells						
River leakage						
Recharge						
Total outflow						
Inflow – outflow						

Model PS3B	**Change in flow rate**[1]				
Budget component	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow					
Storage-specific storage					
Storage-specific yield					
Wells					
River leakage					
Recharge					
Total inflow change					
Outflow					
Storage-specific storage					
Storage-specific yield					
Wells					
River leakage					
Recharge					
Total outflow change					
Inflow – outflow					

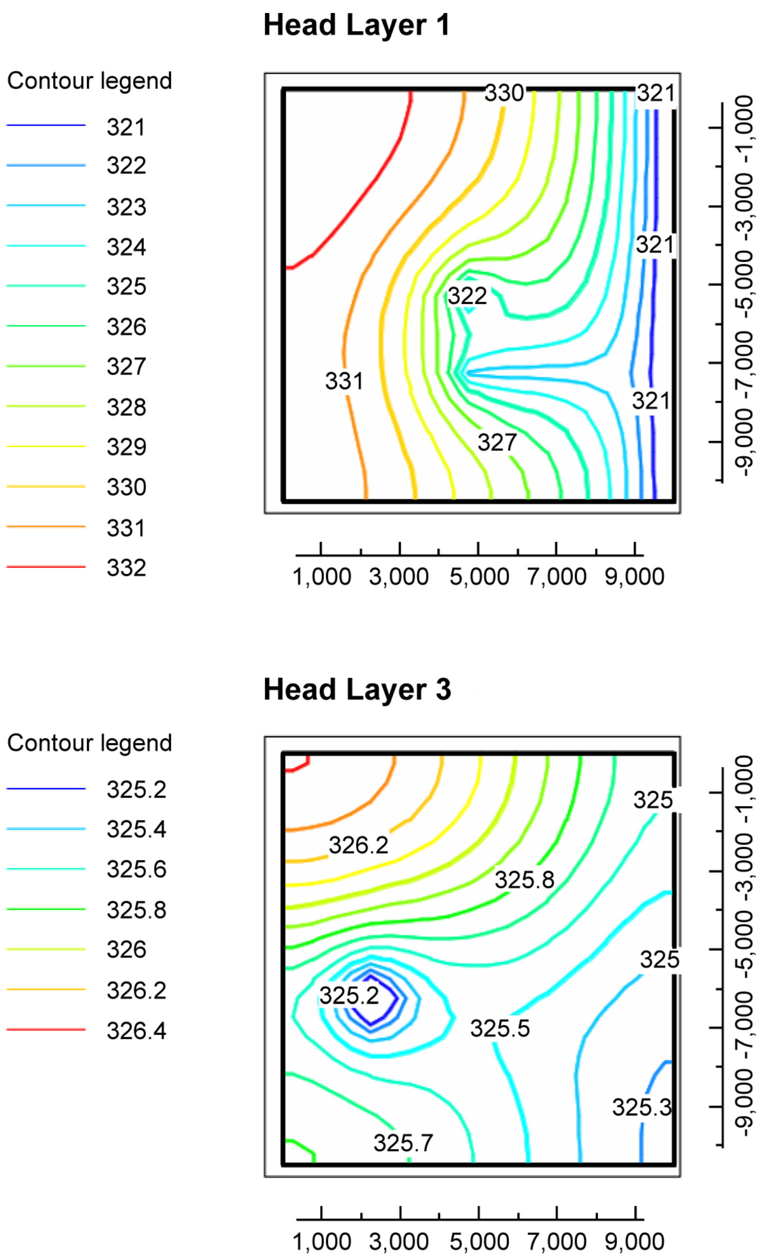
[1] Rate at the current time step minus the initial rate at period 1, step 1. Round to the nearest whole number.

Solution to Exercise 6.4↴

Return to where text links to Exercise 6↥

Exercise 7

MODPATH is a particle tracking program designed to work with output from MODFLOW. This exercise illustrates the major features of MODPATH by applying particle tracking to the final run from Exercise 5.14 (model PS2D1) which included wells in layer 1 and 3 as well as a drain and river in layer 1. The system is relatively complex due to the multiple discharge features. It provides an example of how particle tracking can be used to facilitate visualization of three-dimensional, flow in a complex system. The image below shows the head contours for layers 1 and 3.



Head contours in layers 1 and 3.

[Return to where text links to Exercise 7](#)

Introduction to MODPATH

Exercise 5.14 involved sketching the source area for the well in layer 3. The complexity and variation in the head patterns in layers 1 and 3 make it difficult to provide a quantitative estimate of the source area. In this exercise, particle tracking is used to help delineate the source area and illustrate the major features of the flow system.

Rather than build the needed files, the PSMP files can be extracted from the zip file provided on the book page Getting-Started-With-MODFLOW_Exercises.zip↗. The zip file contains the ModelMuse files for all the completed MODFLOW exercises presented thus far in this book, as well as a file for the PSMP model without the MODPATH component. The file PSMP.gpt can be opened in ModelMuse or, if you already set up a version of the model in another Graphical User Interface, that file can be opened. The model includes only a single steady-state stress period.

Return to where text links to Exercise 7↥

Modify the PSMP Dataset so that it is Ready to Run MODPATH

Open PSMP dataset in ModelMuse or your GUI and follow the steps 1 through 3 to prepare it to run MODPATH. If you have any difficulties following the steps, a link is provided after step five that leads to a box with more detailed instructions.

1. If not already installed, download MODPATH version 7 from the USGS web site and install it by extracting it to an empty directory.
2. Activate MODPATH 7 and set the STOPOPTION to stop the particles at the termination points.
3. Set the porosity for all layers to 0.3.
4. Make sure that the heads and flows will be saved because they are part of the input for MODPATH. In transient models, heads and flows are required for every time step.
5. The dataset should now be ready to run MODFLOW and MODPATH. However, there is one very important step required to make sure things are set up properly. When MODPATH is activated, an auxiliary variable named IFACE needs to be assigned to each entry in the river, drain, well, and perhaps the recharge packages. Your GUI may do this for you. IFACE is the integer boundary face flag that tells MODPATH where within a cell to assign the flow to, or from, those boundaries. By default, IFACE will be treated as if it was zero for all entries. A value of zero indicates that the stress component is distributed internally within the cell and not assigned to a cell face to represent a boundary flow. In this problem, the conceptual model of the river and drain is that they are shallow features located near the top of layer 1. To represent that, the IFACE values for the drain and river cells need to be assigned a value of 6, so MODPATH will assign those flow rates to the top faces of the cells. Recharge is also conceptualized as occurring at the surface, so it also needs to be assigned an IFACE

value of 6. Once those changes are made, the dataset is ready to be used for a MODPATH simulation.

Step-by-step instructions for using ModelMuse to set up the model to use MODPATH are provided in Box 20⤵.

Remember that it is necessary to run MODFLOW again after making these changes so that the budget output required by MODPATH includes data for all stress periods as well as the correct IFACE values before any MODPATH simulations are made. Save this version of the model. It will be the starting point for each of the simulations conducted in Exercises 7.1 through 7.3.

Exercise 7.1

Forward Endpoint Simulation

If it is not already open, open the version of PSMP that was saved after the steps undertaken in Box 20. Save it with a new file name such as PSMP_Endpoints. Then follow steps 1 through 5 below. A link to step-by-step instructions for each of the 5 steps is provided after step 5.

1. Make a forward-tracking endpoint simulation to delineate the source areas for the wells, drain, and river.
2. Place a 5×5 array of particles on face 6 (top face) of each cell in layer 1.
3. Change the setting in MODPATH to read zone arrays.
4. Set the zone numbers of layer 1 and layer 3 so that the river is zone 2, the well in layer 1 is zone 3, the drain in layer 1 is zone 4, and the well in layer 3 is zone 5.
5. Run MODPATH and plot the endpoints at their starting locations with the particles colored by their final zone number.

See the step-by-step instructions in Box 21⤵.

How well do the MODPATH results compare with what you anticipated after running model PS2D1? To view the results again after the next simulation, save them with a different file name.

Solution to Exercise 7.1⤵

Exercise 7.2

Forward Pathline Simulation

Open the version of PSMP that was saved before starting the endpoint simulation. Save it with a new file name such as PSMP_Pathlines. In this exercise, one particle is placed on the top face of layer 1 (face 6) at X= 1750 and Y = 0 through -10,500 (column 4 and rows 1 through 21). Make a pathline simulation and view the results in a GUI.

Follow the steps 1 through 4 to create a new simulation file. A link to step-by-step instructions for using ModelMuse to accomplish the 4 steps is provided after step 4.

1. Set the MODPATH simulation type to be Pathline.

2. Specify the starting locations for particles in layer 1, column 4 from row 1 through row 21. There should be 1 particle per cell placed in the center of the top surface of the cell.
3. Run MODPATH.
4. After MODPATH finishes, open the pathline file in the GUI to view the results.

Step-by-step instructions are provided in Box 22↴.

To view the results again after creating input files and running the next simulation, save them with a different file name.

Solution to Exercise 7.2↴

Exercise 7.3

Forward Time Series Simulation

In this exercise a 5×5 array of particles is placed on the top face of layer 1 (face 6) for a block of cells in the range of row 8 through row 13 and column 4 through column 7. A timeseries simulation is generated in which particle locations are output to a timeseries file at a fixed time interval of 100 days for 300 time points. Follow steps 1 through 4 to create a new simulation file.

1. Open the version of PSMP that was saved before starting the endpoint simulation. Save it with a new file name such as PSMP_TimeSeries.
2. Set the simulation type to Timeseries.
3. Define the time points for the timeseries simulation by specifying fixed time increments of 100 days and a total of 300 time points.
4. Set the starting locations of the particles to be from layer 1, row 8, column 4 to layer 1, row 13, column 7. Make sure that a 5×5 array of particles are specified for the top face (face = 6).

Run MODPATH. After MODPATH finishes, open the time-series file in the GUI to view the results. Try animating the particle movement by selecting the times at which the particles are displayed.

Step-by-step instructions are provided in Box 23↴.

Solution to Exercise 7.3↴

Return to where text links to Exercise 7↥

Exercise 8

Exercise 8.1

Alluvial Aquifer Problem Set

This exercise is a modified version of an exercise provided by a reviewer of this book, Eve Kuniansky, who developed it based on a report by Torres-González and others (2002). It is estimated to take about 3 hours to complete this exercise. Information about the geology and hydrology of the alluvial aquifer under study, and the available site data, is discussed in *Site Information for Exercise 8* at the end of this exercise.

Overall Project Goal

The goal of the site model is to find a location for a well that can withdraw up to 1000 m^3/day from the alluvial aquifer without causing the well to go dry. The alluvial aquifer was deposited in a valley incised into bedrock and hydraulically connected to the Rio de Manati, a river in Puerto Rico. The alluvium will be used to filter the river water, providing pre-treatment. Thus, travel time from the river to the well must be at least 60 days as required to eliminate surface-water borne pathogens. As a safety margin, a travel-time of at least 60 days is preferred. (The recommended travel time might be different for other locations.) MODPATH will be used to estimate travel time from the river to the well. Another consideration with final well placement is locating the well in an area that is rarely inundated.

There are two phases to the problem that reinforce what has already been presented but provide an opportunity for readers to gain practical experience with incorporating field data into a model and as well as other challenges faced in applying a model to a field system. Site Information for Exercise 8↴ is provided as the last subsection of this exercise, including a description of the study area and site data needed for the exercise. The data and images for this exercise can be downloaded from the Groundwater Project web site↗.

Phase 1 of the exercise begins by developing a conceptual model of the alluvial aquifer then building and calibrating the initial model. It is estimated to require 2.5 hours of time to complete Phase 1. In this phase the reader uses the information collected in 1998 to characterize the system, then builds and calibrates a groundwater model using the depth to bedrock data, slug test data, steady-state water level map collected immediately before a step-drawdown test, the step-drawdown test data and analysis of the test conducted by the well driller.

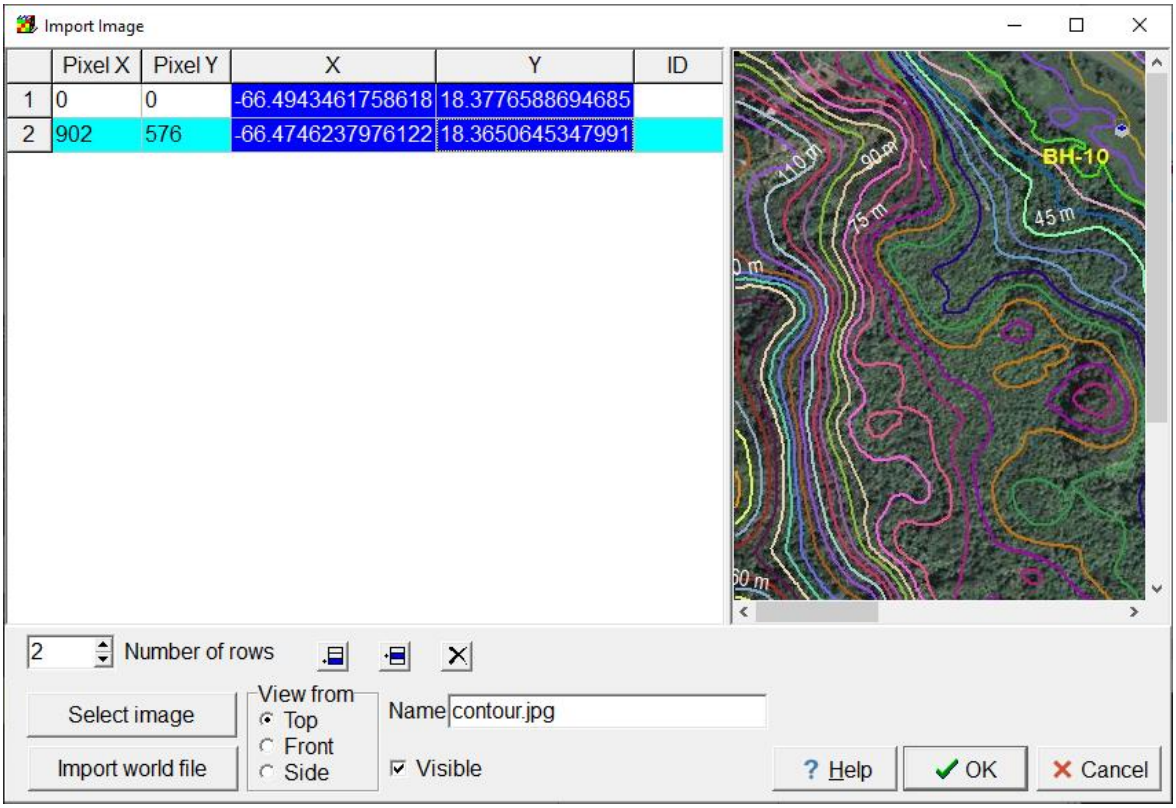
Phase 2 of the exercise uses the model to locate a well outside of the floodplain that will have a 60-day time of travel between the well and the river using MODPATH and confirm that it will not go dry at a pumping rate of 1000 m^3/d using MODFLOW. It is estimated to require 0.5 hours of time to complete Phase 2.

Starting ModelMuse and Importing a Background Image

- Start ModelMuse.
- Set the number of layers to 1 and change the name of the aquifer.
- Retain the default settings for the bottom elevations.
- Click the *No grid* button.
- Select *File|Import|Image…*
- Click the *Select Image* button and select *contour.jpg*.

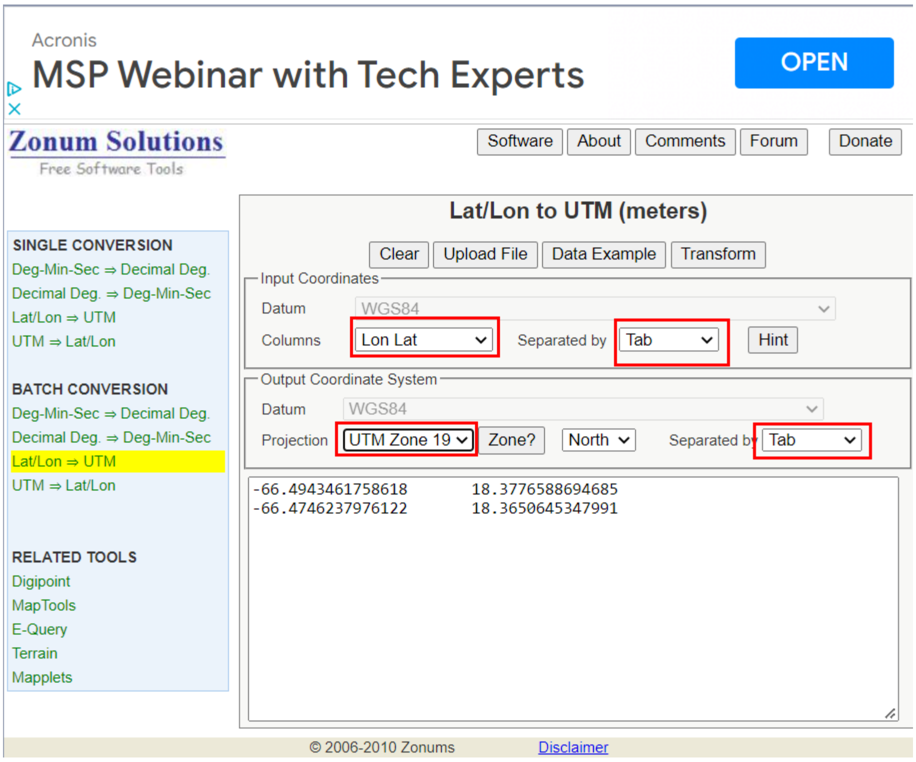
There is a .jgw file associated with contour.jpg that has georeferenced information on the WGS84 latitude and longitude and increment for each pixel in the .jpg. ModelMuse uses this to determine the coordinates of the upper left and lower right corners of the image. However, the coordinates are in decimal degrees for the WGS84 datum. Decimal degrees are not suitable for modeling. Thus, decimal degrees will be converted to Universal Transverse Mercator coordinates UTM Zone 20N. This is a sufficiently small area that the UTM projection will not result in distortion of the site features.

- As shown below, select the block of cells that contain the X and Y coordinates (Longitude, Latitude) in the dialog box shown below. Multiple cells can be selected by clicking on one of the cells and then holding down the shift key while clicking on another cell. Next, press *Ctrl-C* on the keyboard to copy the cells to the clipboard.



Settings on web page for converting coordinates

- Go to http://www.zonums.com/online/coords/cotrans.php?module=13↗. This web page can be used to convert coordinates from decimal degrees to UTM coordinates. By default, it expects the coordinates to be latitude (Y) and longitude (X) in that order. The coordinates copied from ModelMuse have longitude followed by latitude. On the web page, use the Columns drop-down menu to specify the data in longitude followed by latitude. The data are separated by tab characters and the model area is in UTM Zone 19 north. Have the output data separated by tabs. Enter that data on the web page. Once all the data items have been set correctly as illustrated below, click the *Transform* button. The transformed coordinates will appear in a separate window.



Settings on web page for converting coordinates

- Copy the transformed coordinates from the web page and paste into the Import Image dialog box in ModelMuse replacing the original decimal degrees coordinates (select the data and right-click to copy the data). Click *OK*.
- Now it is possible to zoom in on the image.

Another site that can be used for coordinate transformation is the Multipoint Conversion tab on https://www.ngs.noaa.gov/NCAT/↗.

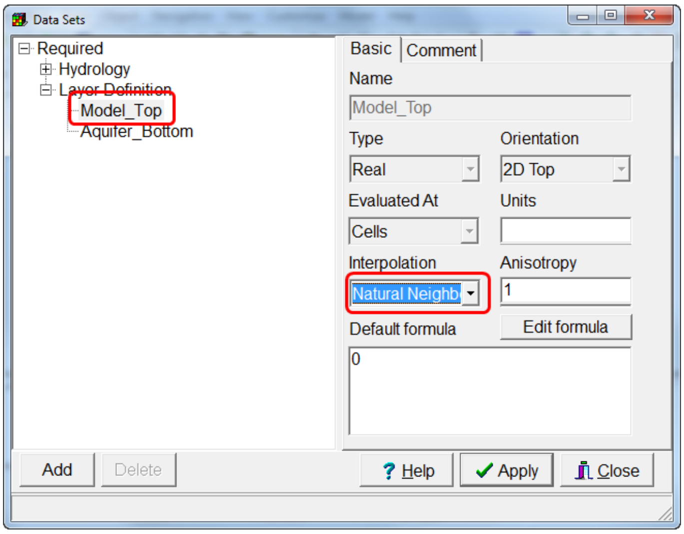
Importing XYZ Data of the Ground Surface

The file riverarea.xyz contains coordinates of points where the elevation of the ground is known. The coordinates are in decimal degrees (WGS84). These data points can be imported into ModelMuse to specify the top of the model after the geographic coordinates are converted to Cartesian coordinates.

1. Open riverarea.xyz in a text editor. Copy all the data to the clipboard and paste it into a spreadsheet.
2. Separate the data so that the coordinates and elevations are in separate columns. The last column is the ground elevation (longitude, latitude, elevation, WGS84 datum). In Microsoft Excel, the function *Data|Text to Columns* can be used to separate the data.
3. Use http://www.zonums.com/online/coords/cotrans.php?module=13↗ to transform the coordinates into the correct UTM coordinates.
4. Copy the converted coordinates to the spreadsheet and align them with the elevations.

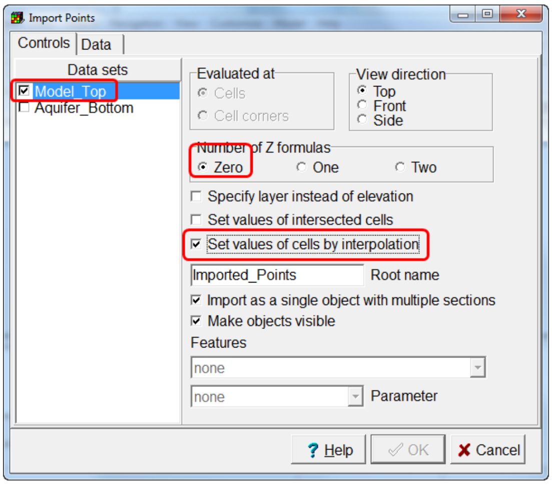
In ModelMuse, interpolation of the data points will specify the elevation of Model_Top. To do that, first specify which interpolation algorithm will be used.

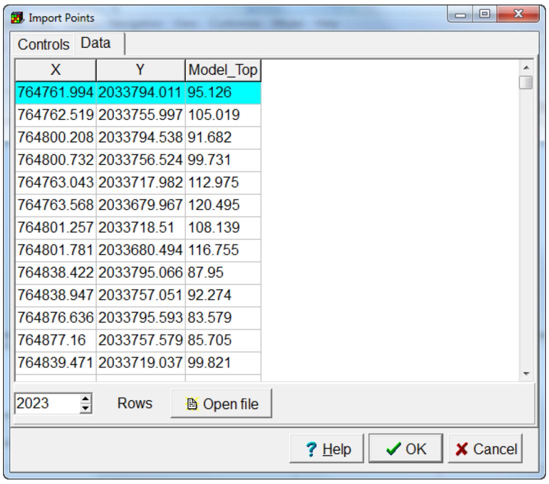
5. As shown below, select *Data|Edit Data Sets…* and then select the *Model_Top* data set. Set the interpolation method to *Natural Neighbor*. Click the *Apply* button.



Assigning Natural Neighbor interpolation method

6. Next import the data points. Select *File|Import|Points…* Check the checkboxes for "Model_Top" and "Set values by interpolation" as illustrated below. Copy the data from the spreadsheet and paste it into the table on the Data tab. Click *OK*. The object that is imported may obscure the background image. It is an array of small squares representing the data points. If so, you can hide the object.

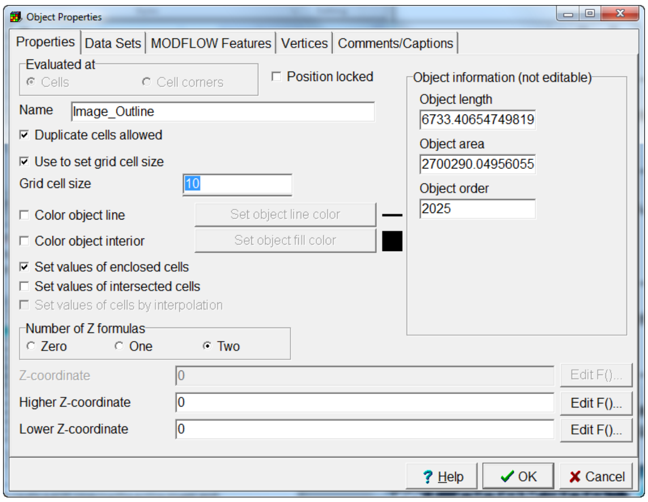




Import Points dialog box with settings and data

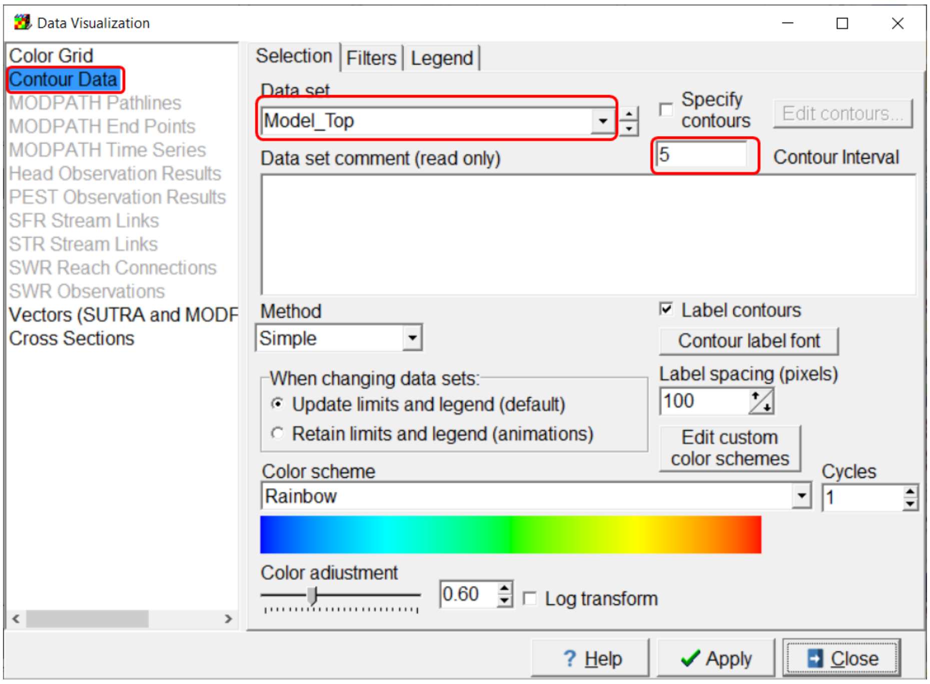
To test that the data were imported correctly, create a grid covering the entire image and use ModelMuse to create contours of the Model_Top data set as explained in the following steps. Then, compare those elevations to the ones in the background image.

1. Create a polygon object that roughly matches the area of the background image. Use it to specify a grid size of 10 as shown below. Click *OK*.



Setting grid cell size

2. Select *Grid|Generate Grid*. Keep all the default settings and click *OK*.
3. Select *View|show or Hide 2D Grid|Show Exterior*.
4. The vertical exaggeration is too high so select *View|Vertical Exaggeration…* and click the *Default* and *OK* buttons.
5. Select *Data|Data Visualization* and select *Contour Data* as shown below. Then select the Model_Top data set, set the contour interval to 5 and click *Apply*. Compare the contours generated by ModelMuse with those embedded in the background image. The contours should be similar but will not be identical.



Contouring the top elevation

Importing Image Containing Domain Outline

Import another background image from the file named "siteswalluvium.png." This image covers a larger area and has a higher resolution. It contains an outline of the extent of the alluvium. Use the extent of the alluvium as the outline of model area. The image contains the locations of a number of boreholes and other features. The coordinates of these features are in SiteInformation.xlsx. They need to be converted to UTM coordinates.

1. Open SiteInformation_T1.xlsx and convert all the coordinates to UTM coordinates.
2. Select *File|Import|Image* and select the siteswalluvium.png. Click *OK* in the Information dialog box that appears. Enlarge the Import Image window so you can view a larger area. In the image, sequentially click on the locations of several of the boreholes or wells. Enter the coordinates from the SiteInformation.xlsx spreadsheet. Some useful boreholes to pick include BH-5, BH-10, BH-6, and BH-1. (The locations on the image are only approximate. For example, well TW2 is

located northeast of well TW4 according to it's coordinates but is shown as being northwest or well TW4 in the image. The coordinates are believed to be more accurate than the image.)

3. Click *OK*.

Create a New Grid of Just the Area Covered by Alluvium

1. Either delete the object that was previously used to specify the domain outline or edit it in the Object Properties dialog box and uncheck the check box for specifying the grid size.
2. Create a new polygon object by tracing the orange line that represents the edge of the alluvium in siteswalluvium.png. In the Object Properties dialog box, check the *Use to set grid cell size* box and set the grid cell size to 10. Click *OK* to close the dialog box.
3. Select *Grid|Generate Grid* to create a new grid.

Check the Active Area of the Grid

By default, all the cells inside the domain outline will be active.

- To check that the active cells have been specified correctly, select *View|Show or Hide 2D Grid|Show Active*.

If the active area needed to be adjusted, it could be done by using objects to specify values for the data set named "Active."

Other Options

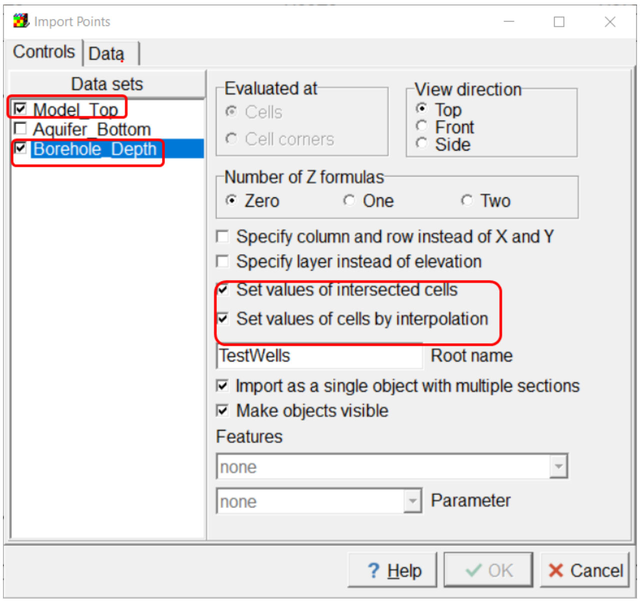
1. Make the aquifer an unconfined aquifer by changing the value of the Cell_Type data set. The Comment in the Data Sets dialog box indicates that if Cell_Type is 0, the cell thickness is zero and if it is greater than zero the cell thickness is calculated. This can be used to determine an appropriate value for Cell_Type.
2. Set the units of the model to meters and days in the *Model|MODFLOW Options* dialog box.
3. Set the initial head to the elevation of the top of the aquifer, using the Data Sets dialog box.

Specifying the Elevation of the Aquifer Bottom

SiteInformation.xlsx contains data on the borehole refusal depth for the boreholes and wells. This will be assumed to be the bottom of the alluvial aquifer. The wells also have data on land surface elevation. These data will be imported and used to compute the elevation of the bottom of the alluvial aquifer.

1. First, define a data set for the borehole depth. Select *Data|Edit Data Sets* and click the *Add* button. Change the name of the new data set to "Borehole_Depth."
2. Now import the borehole depths and the surface elevation for the wells as shown below. Select *File|Import|Points…* and check the boxes for the "Model_Top"

and “Borehole_Depth”. Also check the boxes for “Set values of intersected cells” and “Set values of cells by interpolation”. Specify the name of the object to be imported (the Root name) as “TestWells.” On the Data tab, copy and paste the coordinates of the well, the land surface in meters (as Model_Top) and the borehole refusal depth in meters. Click *OK*. There will be a warning that interpolation is not used on one of the data sets. In this case, the Borehole_Depth data set does not use interpolation. However, it will not be a problem so choose *Yes* in the dialog box. The data points will be interpolated for the “Model_Top” data set.



Import Points dialog box

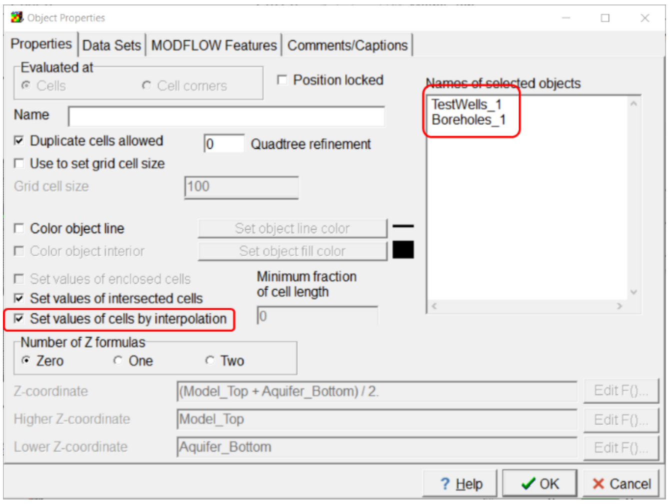
3. Repeat these import steps for the boreholes but only import the borehole depth. Do not check the box for “Set values of cells by interpolation.” During or after importing the data, change the name of the new object to have the root name “Boreholes.”

4. Calculate the elevation of the aquifer bottom as Model_Top - Borehole_Depth. However, do not use a default formula. Instead, calculate the value at each well and borehole and interpolate between those values. First, in the Data Sets dialog box, specify the interpolation method of the Aquifer_Bottom data set as Natural Neighbor.

5. Next, change the values of both the TestWells and Boreholes objects so that they set the value of the aquifer bottom. Select both the TestWells and Boreholes objects. To select more than one object, click on the first one in the top view of the model to select it. Then hold down the shift key on the keyboard and double-click the other object.

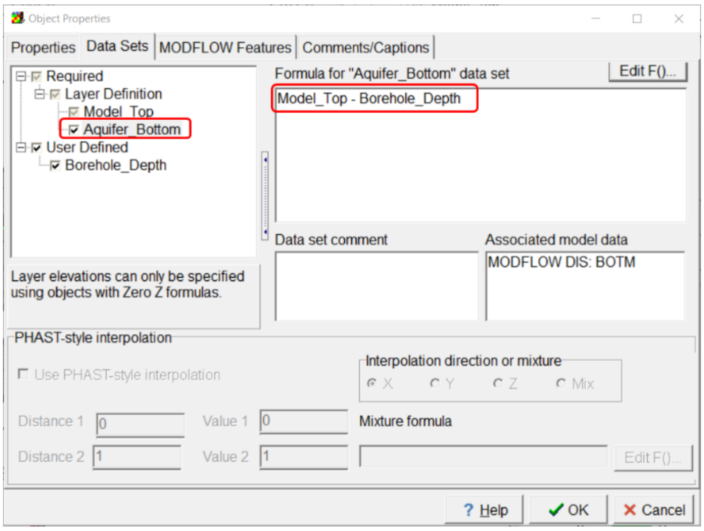
1. In the Object Properties dialog box, the names of the objects will be displayed as shown below. Make sure that the correct objects are being edited. Also note

that the "Set values of cells by interpolation" is grayed. That is because it is specified differently for the two objects. For the Wells object it would be checked and for the Boreholes object it would be unchecked. Check the "Set values of cells by interpolation" box.



Setting values of cells by interpolation

6. On the Data tab, check the box for the Aquifer bottom data set as shown below. Then set its formula to "Model_Top - Borehole_Depth." Note that the "Model_Top" checkbox is grayed. Be sure it remains gray. Click *OK* to close the "Object Properties" dialog box. At this point the "Model_Top" and "Aquifer_Bottom" should both be defined.



Calculating aquifer bottom

7. Experiment with coloring or contouring the grid with the "Model_Top" and then with the "Aquifer_Bottom" data set to make sure the values seem reasonable in the active area of the model.

It is worthwhile reviewing how the aquifer bottom is calculated. The calculation takes place in several steps.

First, the “Model_Top“ data set is calculated. The first step is to interpolate between the XYZ data of the ground surface that was imported and the elevations of the well tops. The “TestWells“ object sets values of cells intersected by the well locations, so in those cells the interpolated value is overridden by the value specified in the object. This is likely to be similar to but not identical to the interpolated value.

Second, the value of the “Borehole_Depth“ data set is calculated. No interpolation method has been assigned to that data set, so the default formula is used to assign a value of 0 to all the cells. Then the values in the “TestWells“ and “Borehole objects“ will be assigned to the cells intersected by those objects.

Next, for each point in the “TestWells“ and “Boreholes objects“, the value of the “Model_Top - Borehole_Depth“ is calculated. The “Boreholes object“ does not set the value of “Model_Top“ so the value it uses in the equation will be an interpolated value. Because the aquifer bottom data set has an interpolator assigned, and because both the Wells and “Boreholes objects“ set values of cells by interpolation, the values for each well and borehole point will be interpolated to define values for every cell. Finally, because the “TestWells“ and “Boreholes objects“ also set values of intersected cells, the values of cells intersected by the objects will be reset by them, so the values assigned to intersected cells might be slightly different from the interpolated values.

Assigning Hydraulic Conductivity

Hydraulic conductivity is assigned in much the same way as elevations are assigned. The one difference is that interpolation cannot be used to assign values to 3D data sets. However, a formula can be used in a 3D data set to link to a 2D data set in which interpolation is allowed and that is described by the following steps.

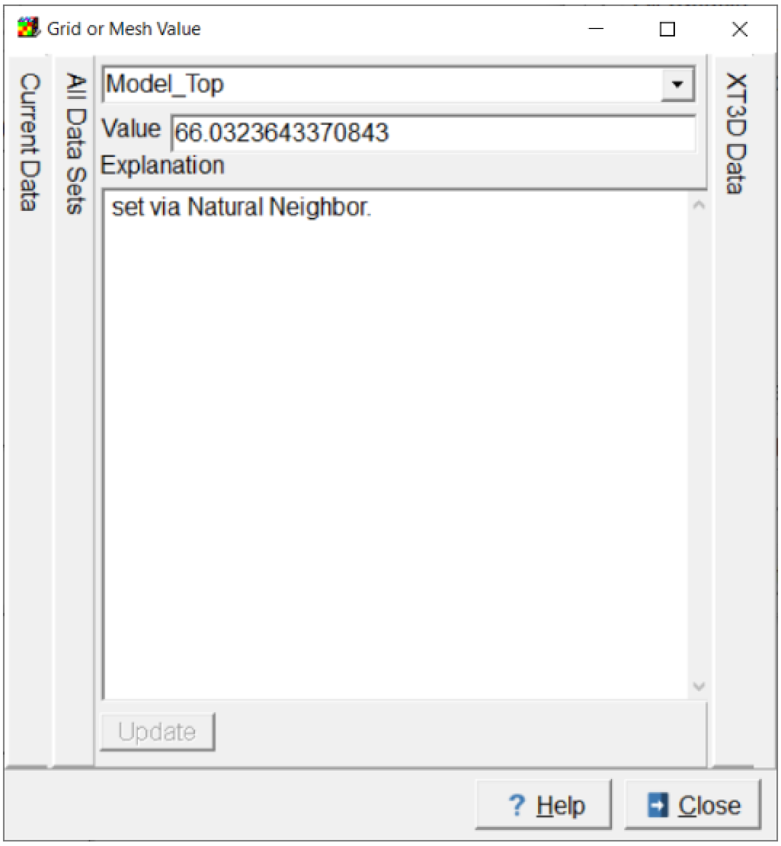
1. Define a data set named “Kx_2D” and set its interpolation method to Natural Neighbor.
2. Slug tests were used to estimate the hydraulic conductivity in the wells. Slug test results are provided in the SiteInformation_T1.xlsx spreadsheet available for download with this book and are also tabulated in the final section of this exercise, *Site Information for Exercise* 8. Import that information and use it to specify the Kx_2D data set by interpolation.
3. Next, select *Data|Edit Data Sets*… and set the formula for the Kx data set to Kx_2D.

Define River Boundary

The river is an important boundary condition for this model. Assume the river bed sediments have the same hydraulic conductivity as the aquifer and the head in the river varies gradually from a higher value in the south to a lower value in the north. Use the elevation of the top of the model to pick a few elevation points along the river and interpolate the elevations between those points along the length of the river as described below.

1. Activate the River package in the MODFLOW Packages and Programs dialog box.

2. Select *Data|Show Grid or Mesh Values* and use the All Data Sets tab to display values of the Model_Top data set. Alternatively, you could create contour lines of the Model_Top data set.

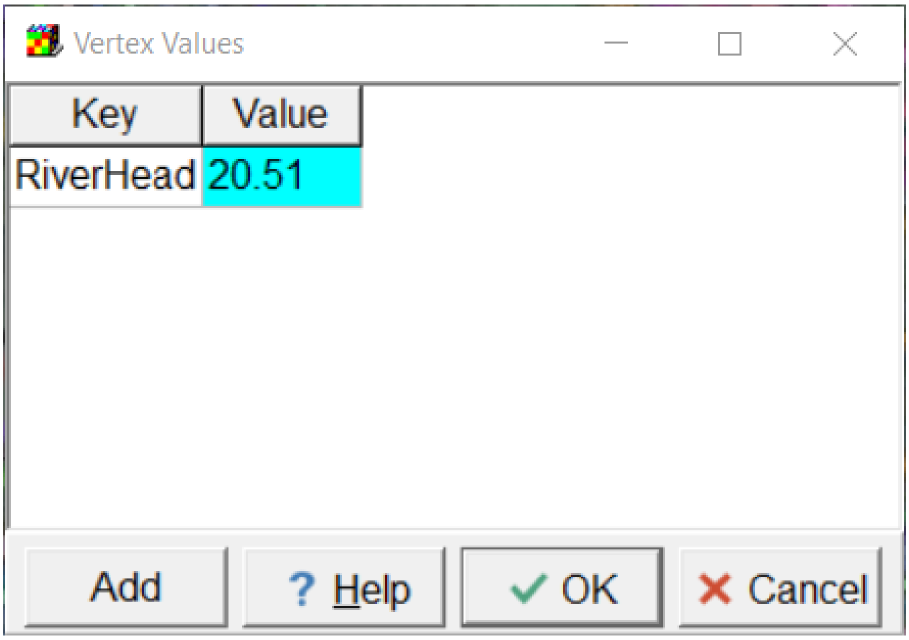


Grid or Mesh Values dialog box

3. Draw a polyline object along the river. You may need to display the background image to see where the river is. For now, do not use the polyline object to set the values of any data sets or Model Features. Be sure to put a vertex near where each River Stage site in the image is located and near the two temporary gages. Close the Object Properties dialog box when you are done without using it to assign any values.

4. After creating the object, select *Object|Edit|Vertex Values*. Then double-click on one of the vertices of the object that is at one of the temporary or permanent river stage sites. The Vertex Values dialog box will appear. Enter “RiverHead” as the key. For the value, enter the river stage value for that site from the Water levels table (Table 4↴) in the final section of Exercise 8 *Site Information for Exercise* 8. Repeat this for several more points along the length of the river. It may be necessary to zoom in to read the site identifier and the vertices may need to be

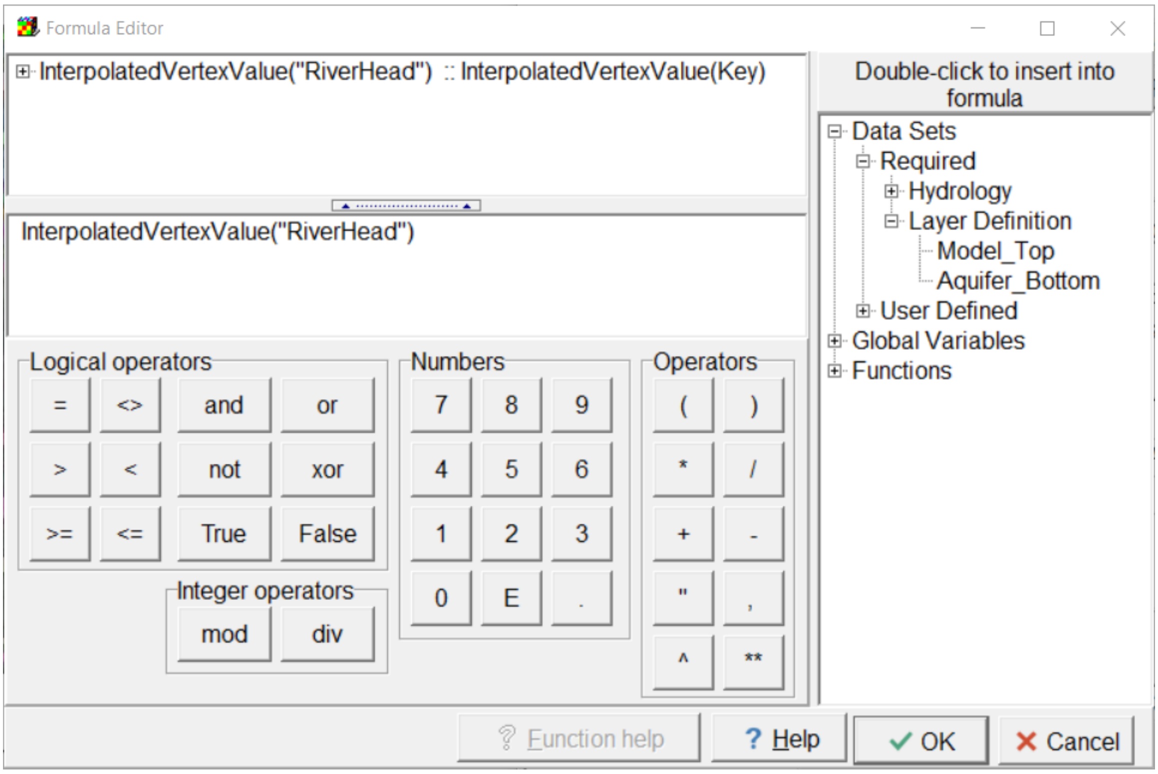
moved closer to the stage-measurement site. You may also select additional points along the river and assign values based on the top of the model. (See step 2.)



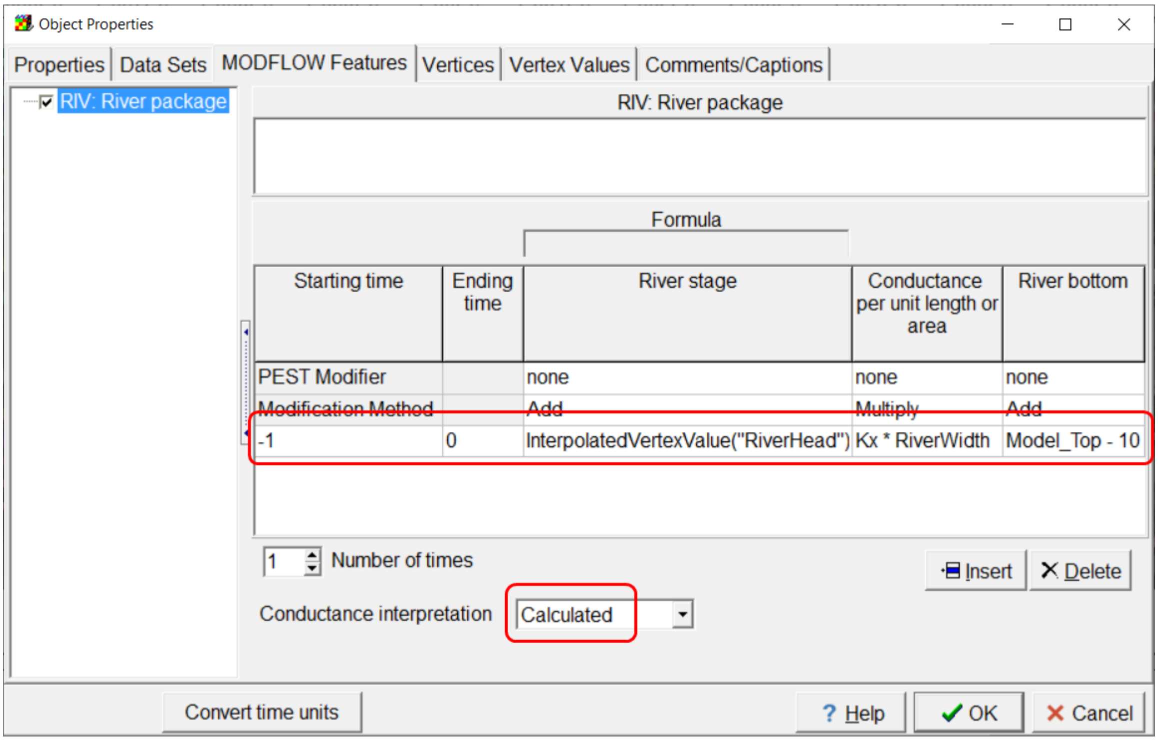
Vertex Values dialog box

5. Use *Navigation|Measure* to measure the width of the river. Then create a global variable named RiverWidth that has the width of the river as its value.

6. Select *Object|Select Objects* and double-click on the river object. In the Object Properties dialog box, under the MODFLOW Features tab, activate the River package. Enter the starting and ending times for the model (-1 and 0 for this steady-state simulation) and then under "River Stage," click on the *F()* button to open the Formula Editor. Locate the *InterpolatedVertexValue* function, select it, and then click *Function help* to read about what the InterpolatedVertexValue function does. You have already entered vertex values associated with "RiverHead." So now use the InterpolatedVertexValue function to assign values of the River Stage (see figure below). From the information provided, the river is well incised into the alluvium, so the bottom of the river bed can be set using the information on the cross-section, provided in the final section of this exercise, Site Information for Exercise 8, as 10 meters below the MODEL_TOP or 10 meters above the bottom. Alternatively, it could be set to 1 meter below the river stage. Then set the conductance interpretation to "Calculated" and assume the river bed thickness is 1 meter so the conductance per unit length of the river bed sediments will be the hydraulic conductivity times the river width.



Assigning river head with the InterpolatedVertexValue function



Assignment of River Properties

7. At this point, enough information has been specified to conduct a steady-state model run. Run the model to see if the simulated water levels seem reasonable. Given the variability in how one sets the river bottom and river stage, and the elevations of some areas of the model, the input may require corrections.

If the Errors and Warnings dialog box appears, check to see if the inputs require correction. Use ListingAnalyst to examine the ".lst" file. Did the model converge? Does the VOLUMETRIC BUDGET show that mass balance is preserved? If everything seems OK, save the ModelMuse file with the name "AlluvialSS." The SS indicates that this is a steady-state model.

Define Observations

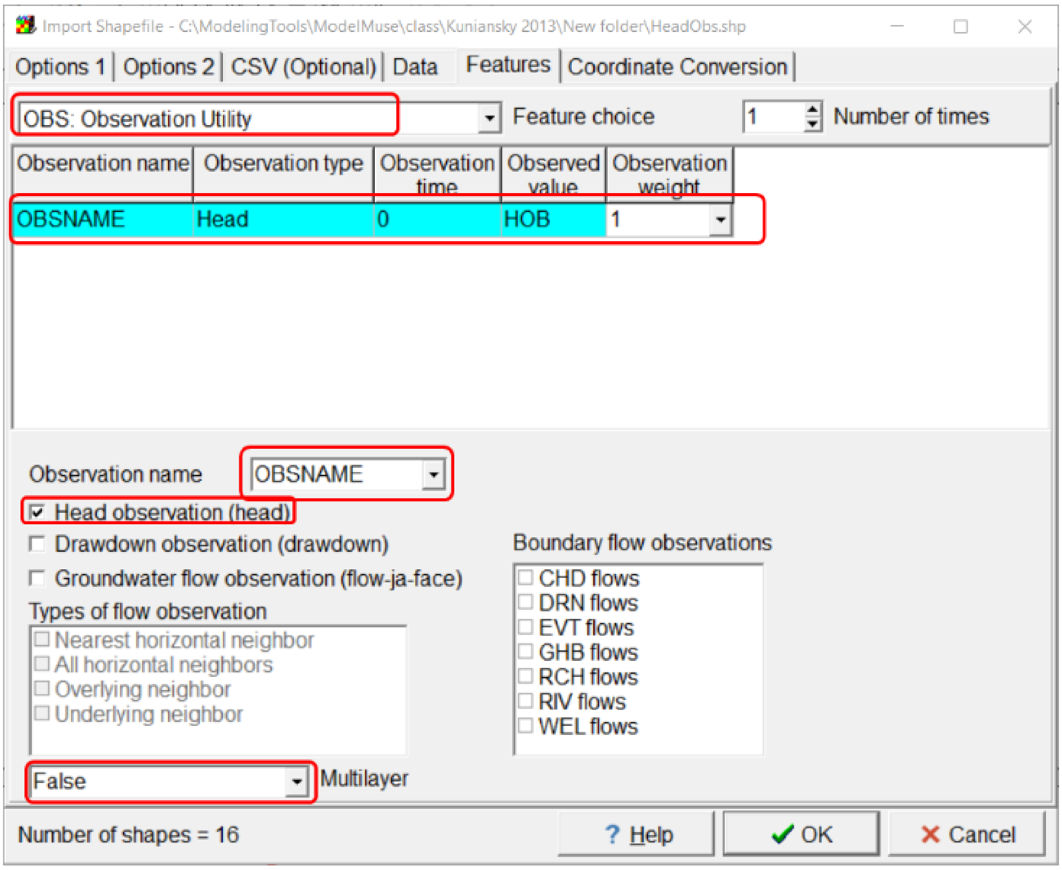
1. Next use ModelMuse to specify head observations. Start by saving the model as AlluvialSSHOB. Entering the Observed Water Levels (heads) in Model Muse facilitates plotting residuals. A residual is an observed minus a simulated value. In this case, a difference in water levels. The data downloaded for this problem includes a Shapefile containing the observed heads and river stages for the model area.

Observed water levels

Identifier	Water level altitude (meters)
TW-1	19.33
TW-2	19.37
TW-3	19.38
TW-4	19.40
TW-5	19.38
TW-6	19.36
TW-7	19.42
TW-8	19.46
RS-2	20.12
RS-3	19.81
RS-4	19.46
RS-5	19.45
RS-6	19.16
RS-7	19.16
SG50036400	18.90
SG50036200	20.51

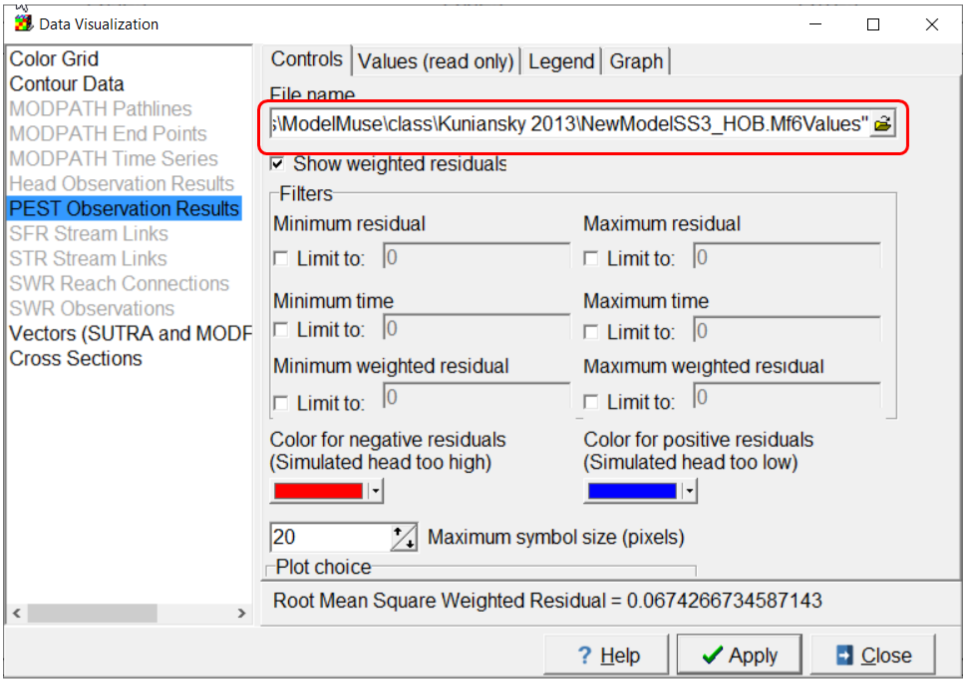
2. First, turn on Observation Utility in the *Model|MODFLOW Packages and Programs* dialog box.
3. ModelMuse has a utility program that works with the parameter estimation program PEST. It can be used to extract simulated values corresponding to the locations and times of the observed values. To use this utility program, PEST must be activated in ModelMuse even though we will not use the PEST software. To activate PEST, select *Model|PEST Properties* and change the PEST Status to *Only define Observations*.
4. Select *File|Import|Shapefile* and select *HeadObs.shp*. Check *Set values of intersected cells*. On the Features tab, select the Observation utility. Use

"OBSNAME" for the observation name in both locations. Select *Head* as the observation type. Set the Observation time to 0, the Observed value to HOB, and the Observation weight to 1. Multilayer should be set to False.



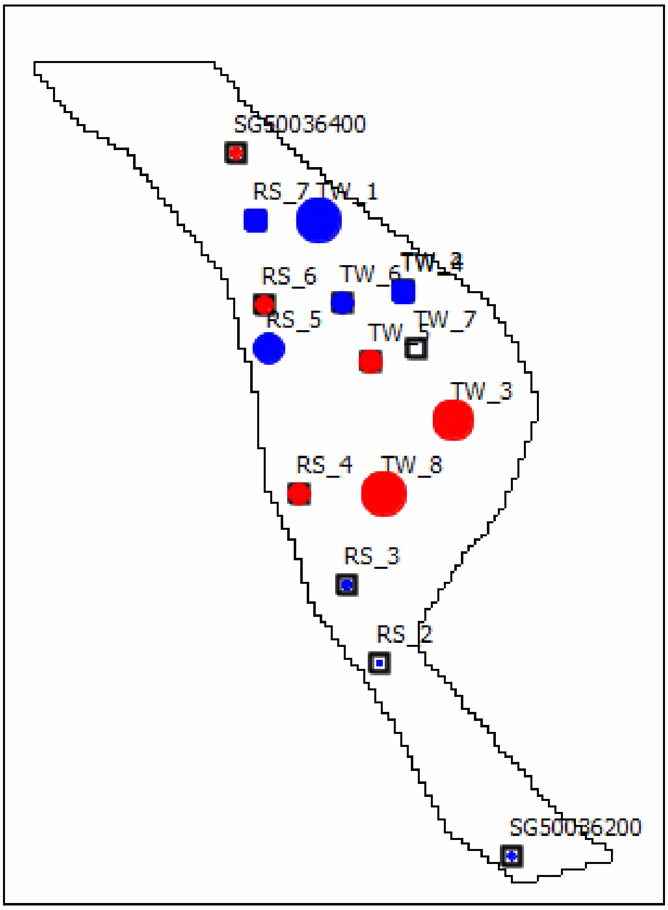
Assigning a head observation

5. Run the model again. The utility program MF6ObsExtractor runs after MODFLOW finishes running. It extracts simulated values at the observation locations for comparison with the observed values. These can be plotted using ModelMuse. Select *Data|Data Visualization* and select *PEST Observation Results*. In the Browse dialog box where the file can be selected, change "files of type" to "Result files" in order to see the file with the extension MF6Values. Select the file with the extension MF6Values.



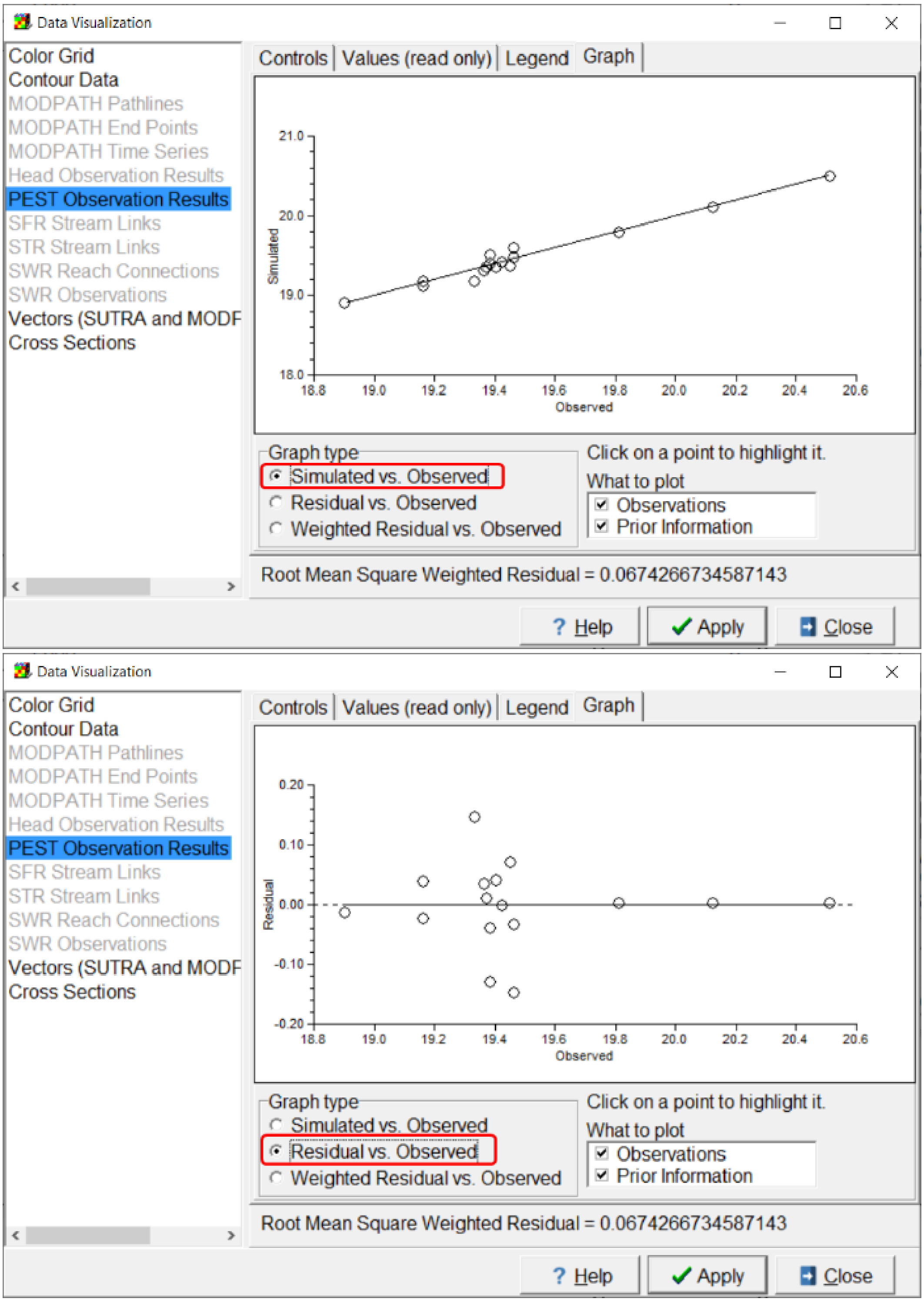
Importing simulated values

6. When you click on *Apply*, the residuals will be displayed in the top view of the model.



Spatial plot of residuals

7. Experiment with the tabs to see graphs of residuals or simulated values versus observed values. The values displayed in the graph may differ slightly depending on how you defined the domain outline.



Simulated Values and Residual Values versus Observed Values in the Data Visualization dialog box

8. Try to improve the model fit by adjusting the hydraulic conductivity distribution. You can do that by adding new objects that define the hydraulic conductivity, removing existing objects, or editing values assigned using existing objects.

Setting up Transient Model to Simulate Initial Condition and Step Drawdown Test

Water levels were collected shortly before the step-drawdown test along with the river stage values during a period when there was not much rain and the river flow and stage were thought to be steady throughout the days before, during, and after the step-drawdown test. Pressure transducers were installed in TW-6, TW-7, and TW-2 and removed at 5pm the next day. The pumping rates from TW-4 are shown in the following table. The transient model will use storage parameters, so set Specific yield to 0.05 for this aquifer.

Timing and Rates of Pumped well TW-4

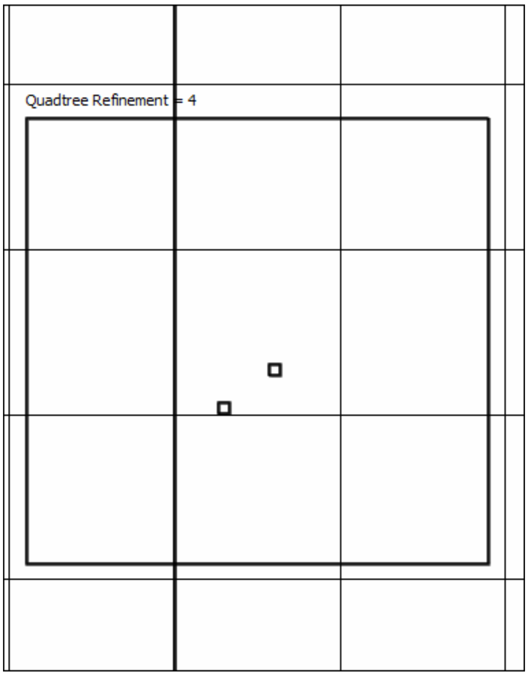
Start time	Rate meter^3/day
8 AM	-1000
11 AM	-1500
2 PM	-2000
5 PM	0

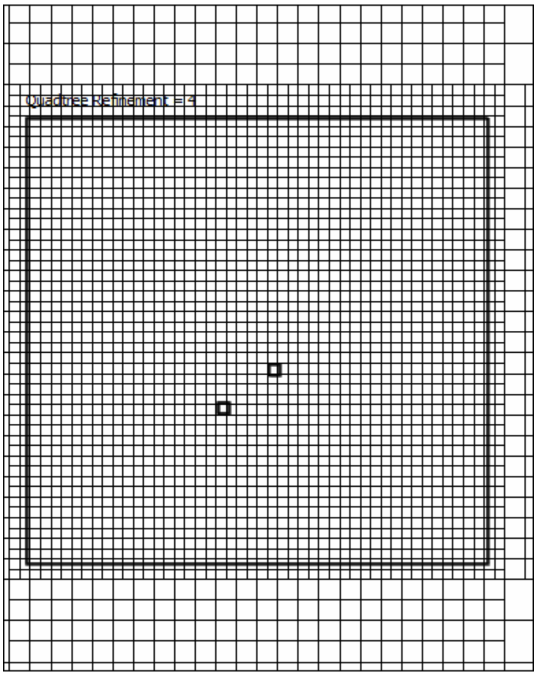
1. Before setting up transient stress periods with pumping, open AlluviumSS and save this as “AlluviumTR.” *Do not start with the ModelMuse file that has head observations.*

2. Go to *Model|MODFLOW Time* menu, to add the required stress periods. Include a steady-state initial period and then add 3 transient pumping periods and a transient period when the pump turns off for a day for a total of 5 stress periods. Calculate the elapsed time in days for the three pumping periods. The length of the last stress period is 1 day. Hint: time zero is the start of the first pumping period and three hours equals 0.125 days. Time step input will be entered later in the exercise.

3. Remember to set the start and end time for the River feature, so it is active for the entire simulation. Double click on that object to bring up the Object Properties window; then set start time for River to -1 and end time to the last total elapsed time (Hint: 1.375).

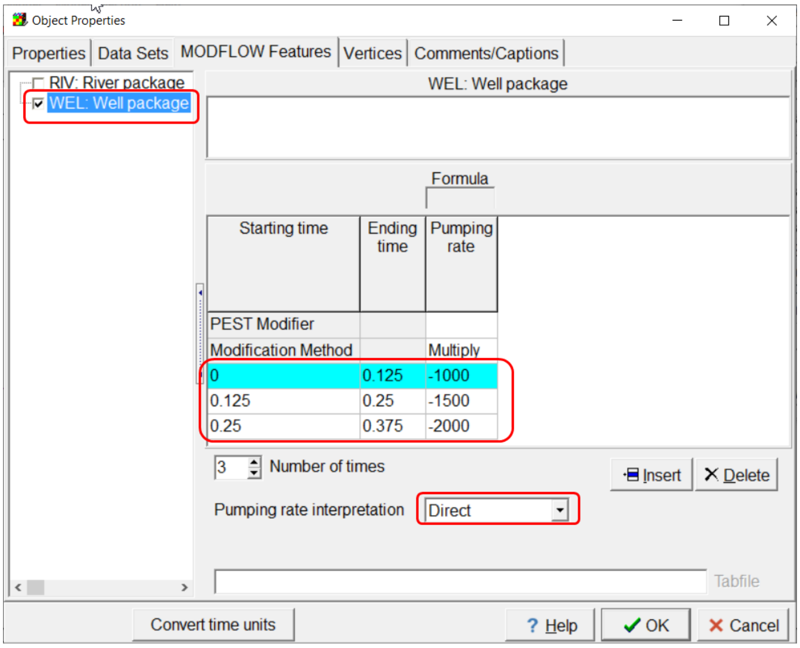
4. Zoom into the area with wells TW-2 and TW-4, they may be in the same cell and if not will certainly be in neighboring cells (this depends on the grid you set up). Ideally, there should be several cells separating the pumping well and the observation well. To achieve this, locally refine the grid around these two wells. To do this, create a rectangle or polygon object surrounding the cells near TW-2 and TW-4 and assign the Quadtree refinement level a value of 4. Then select *Grid|Switch to DISV* to switch to Discretization with vertices. (As mentioned previously, the locations of the wells in the model are based on their coordinates rather their locations shown on sitealluvium.png.)





Application of quadtree refinement (The locations shown in this image are correct based on the provided coordinates. The locations shown in siteswalluvium.png are not precise.)

5. Activate the Well package and create a point object at the location of TW-4 (Test well 4). Name it "PumpingWell." Open it in the Object Properties dialog box and go to the MODFLOW features tab. Select the Well package and enter the pumping rates for each stress period as illustrated below. No pumping rates need to be defined for the steady-state stress period or the last stress period. If you wish, you can enter a pumping rate of zero for those stress periods.



Pumping rates for well test

6. A few solver variables need to be changed to achieve a satisfactory solution with the new model. In the MODFLOW Packages and Programs dialog box, Select the IMS solver and override the default linear acceleration method to specify BICGSTAB. In addition, reduce *Outer DVCLOSE* and *Inner DVCLOSE* to 0.0001 and 0.00001 respectively.

Next, activate PEST observations and specify the head observations at TW-2, TW6, and TW-7 to see if the model is reasonable again before moving to the second part of the exercise. The second part involves determining a location where a production well can pump 1000 m^3/day, in an area that does not get inundated, and has at least 60 days of time for water to travel from the river to the well.

The spreadsheet TR_WL_Obs.xlsx has the transient water level altitudes provided by the drilling company in the format required for reading into ModelMuse to create the HOB file in MODFLOW-2005.

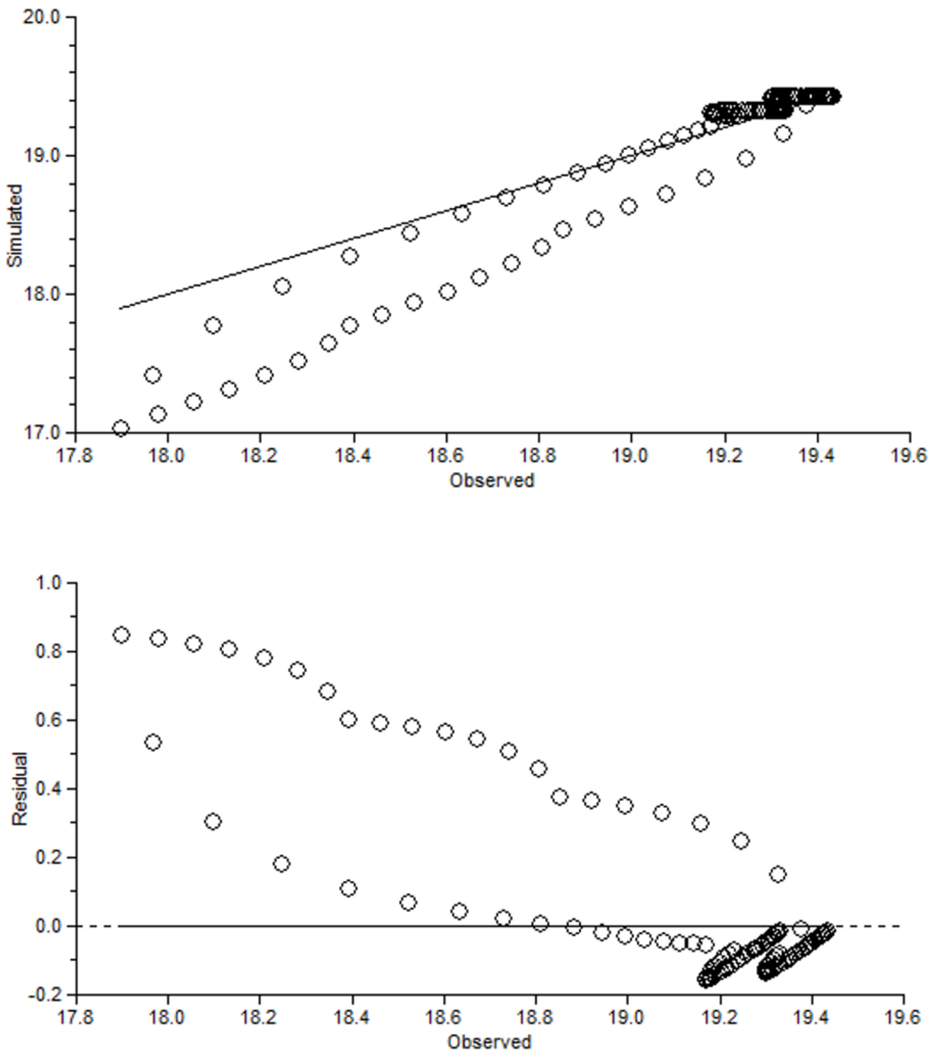
1. Open *Model|MODFLOW Packages and Programs*, expand *Observations* and check *OBS: Observation Utility* to activate it.
2. Open *Model|PEST Properties* and set the PEST status “only define observations.”
3. Create point objects for each of the test wells, enter the observed values for each well, and specify the associated head observations. The user can copy the times and values from the spreadsheet and paste them into the calibration tab for each observation location. The values in the other columns can copied from a spreadsheet in the same way but the user must create those columns in the spreadsheet themselves. Similarly, the user can copy the coordinates of the

observation location and paste them into the vertices tab of the Object Properties dialog box.

4. Go back to the *Model|MODFLOW Time* dialog box and modify the number of time steps in each stress period so that time step sizes will be similar to, or smaller than, the difference in time between observations. This ensures that there will be simulated values close in time to the observed values.

5. Run MODFLOW. In the *Data|Data Visualization* dialog box import the MF6Values file. In general with pump tests, the model does not duplicate the observations exactly.

6. Try to obtain a better fit to the data by adjusting K in the ModelMuse file containing the slug test observations for the points at TW-4 and TW-2. Do not spend too much time on this if you have limited time and want to be sure to complete Phase 2. If you can obtain a better fit, using modified K values, you can modify the steady-state version of the model to incorporate the updated K values.



Simulated Values and Residual Values versus Observed Values in Pump Test.

There is some bias in the fit to the pump test in the model version used to create these graphs, but the RMS of 0.27 is acceptable for continuing the exercise

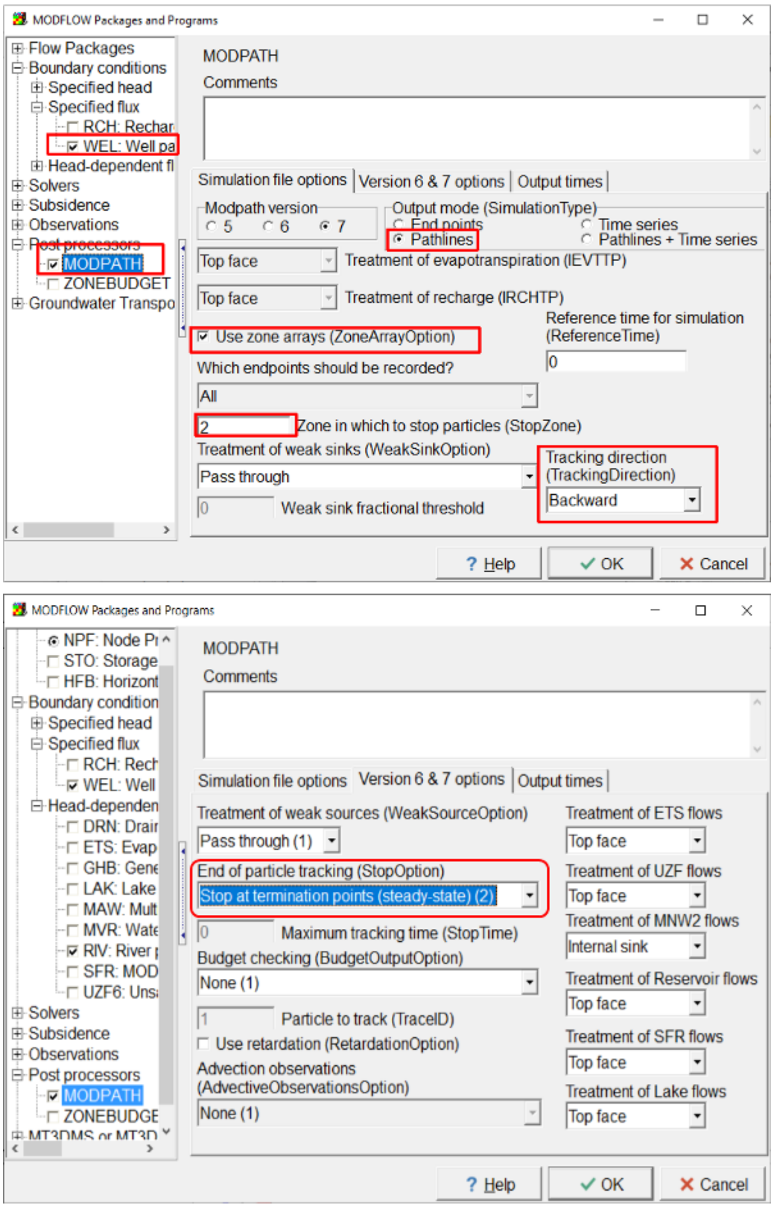
Solution to Exercise 8.1⤵

Exercise 8.2

Determining an Appropriate Location for a Production Well

Set up MODPATH

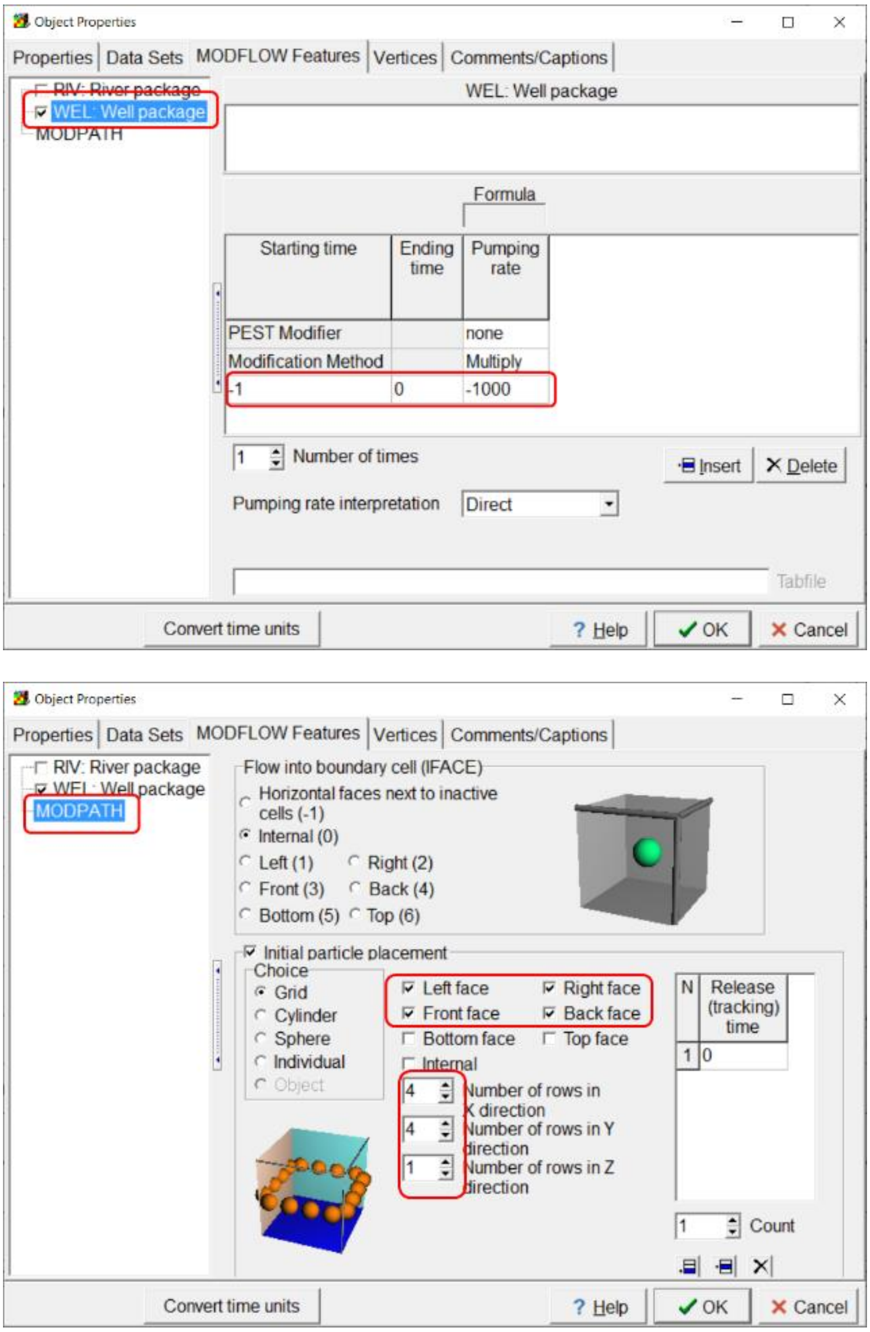
1. Open the steady state model and save it as “AlluvialSS_MPath.”
2. Activate MODPATH and the Well package in the MODFLOW Packages and Programs dialog box.
3. For MODPATH, choose *Pathlines*, *Backwards* particle tracking, *Use zone arrays*, StopZone = 2, and *Stop at termination points*.



MODPATH options

4. The river is the source of water for the well; so set the river's Modpath_Zone data set to 2.

5. Activate the Well Package. Create a point object and use it to define a well with a withdrawal rate of -1000. Also have it be the starting point for the MODPATH particles. Define several particles on each lateral face of the cell. It would be reasonable to start with the well near TW-4. If that site proves unsatisfactory, then move the well and try again.



Point object specifying well and MODPATH starting locations

6. Remember to do a steady-state simulation, but with the pumping well.
7. Decide whether to add long-term recharge to see how the addition of recharge affects particle movement if you have time.
8. Select *Data|Edit Datasets* and then select the Porosity data set. You will see that the default porosity is 0.25. Experiment with different porosity values to explore how they affect travel time.

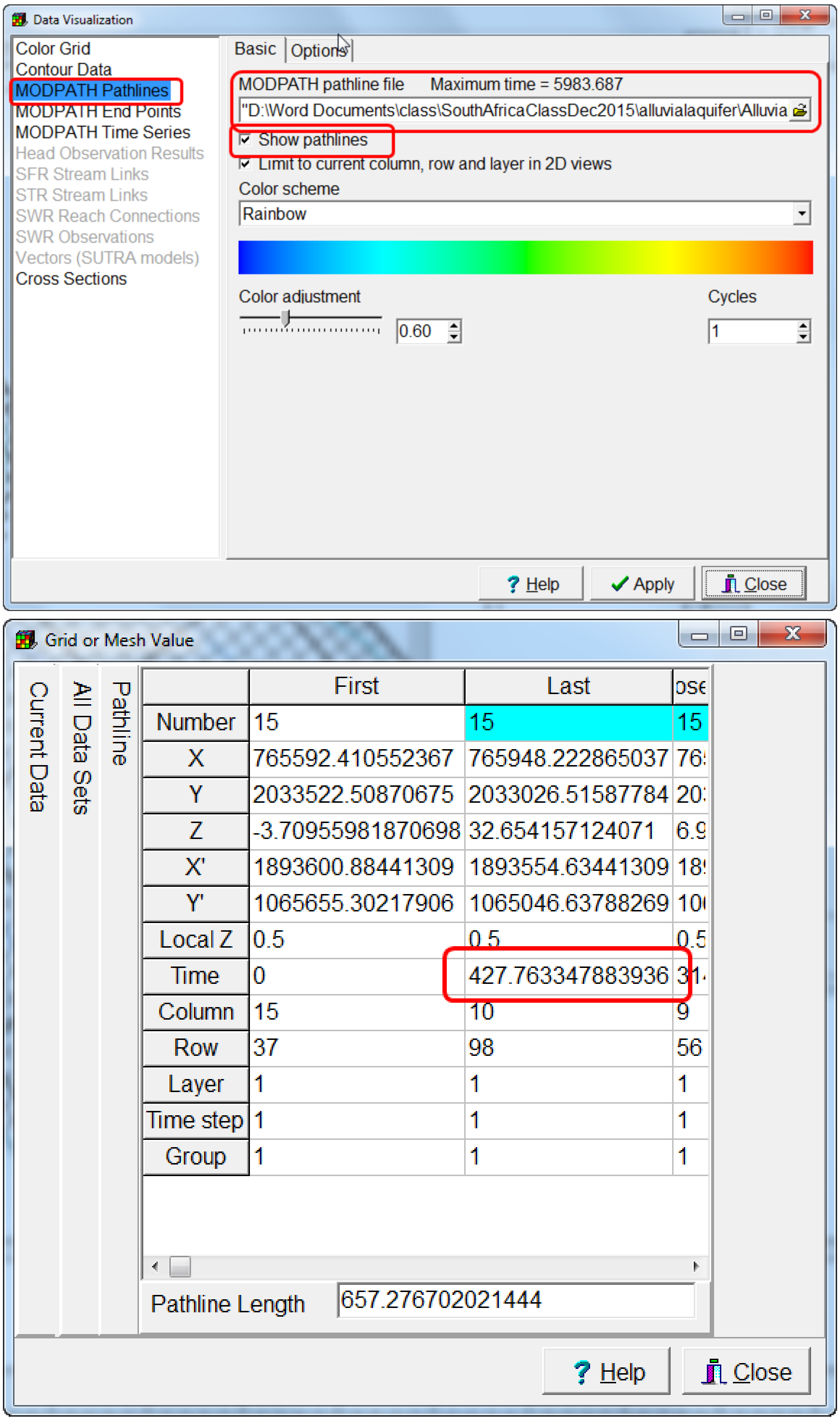
Run the Model and Analyze results

Run the model using *File|Export MODFLOW 6 Input Files* and check the listing file to make sure that there is some flow to the well. If the cell containing the well went dry, the flow to the well will be zero. If that happens, try moving the well to a different location. Make sure the tested locations are in the region that will not be frequently flooded.

```
 VOLUME BUDGET FOR ENTIRE MODEL AT END OF TIME STEP    1, STRESS PERIOD   1
 ---------------------------------------------------------------------------------

    CUMULATIVE VOLUME      L**3       RATES FOR THIS TIME STEP      L**3/T
    ------------------                ------------------------

          IN:                                  IN:
          ---                                  ---
                WEL =          0.0000                WEL =          0.0000
                RIV =       1297.6639                RIV =       1297.6639

           TOTAL IN =       1297.6639           TOTAL IN =       1297.6639

         OUT:                                 OUT:
         ----                                 ----
                WEL =       1000.0000                WEL =       1000.0000
                RIV =        297.7641                RIV =        297.7641

          TOTAL OUT =       1297.7641          TOTAL OUT =       1297.7641

           IN - OUT =         -0.1002           IN - OUT =         -0.1002

PERCENT DISCREPANCY =         -0.01   PERCENT DISCREPANCY =         -0.01
```

MODFLOW budget

Import the MODPATH Pathlines in the Data Visualization Window. Then use the Grid or Mesh Values dialog box to check the travel time from the well backwards to the river. If the travel time is too short, try moving the well to a different location.



Plotting pathlines

See what happens if the particles are removed from the well and instead placed in the river cells with forward tracking simulated. The river should not be defined as a stopping zone when tracking from the river.

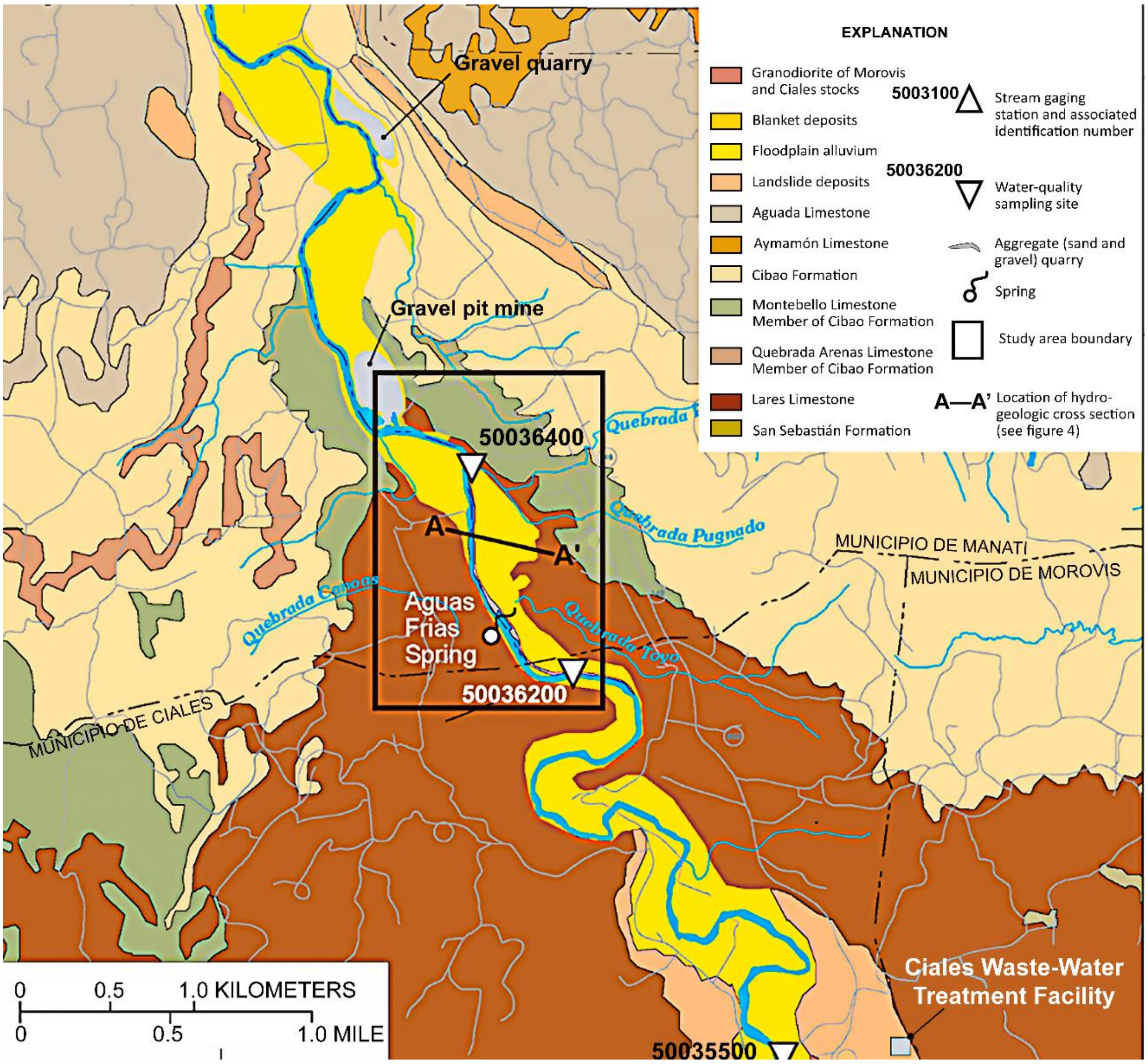
Solution to Exercise 8.2↴

Return to where text links to Exercise 8↥

Site Information for Exercise 8

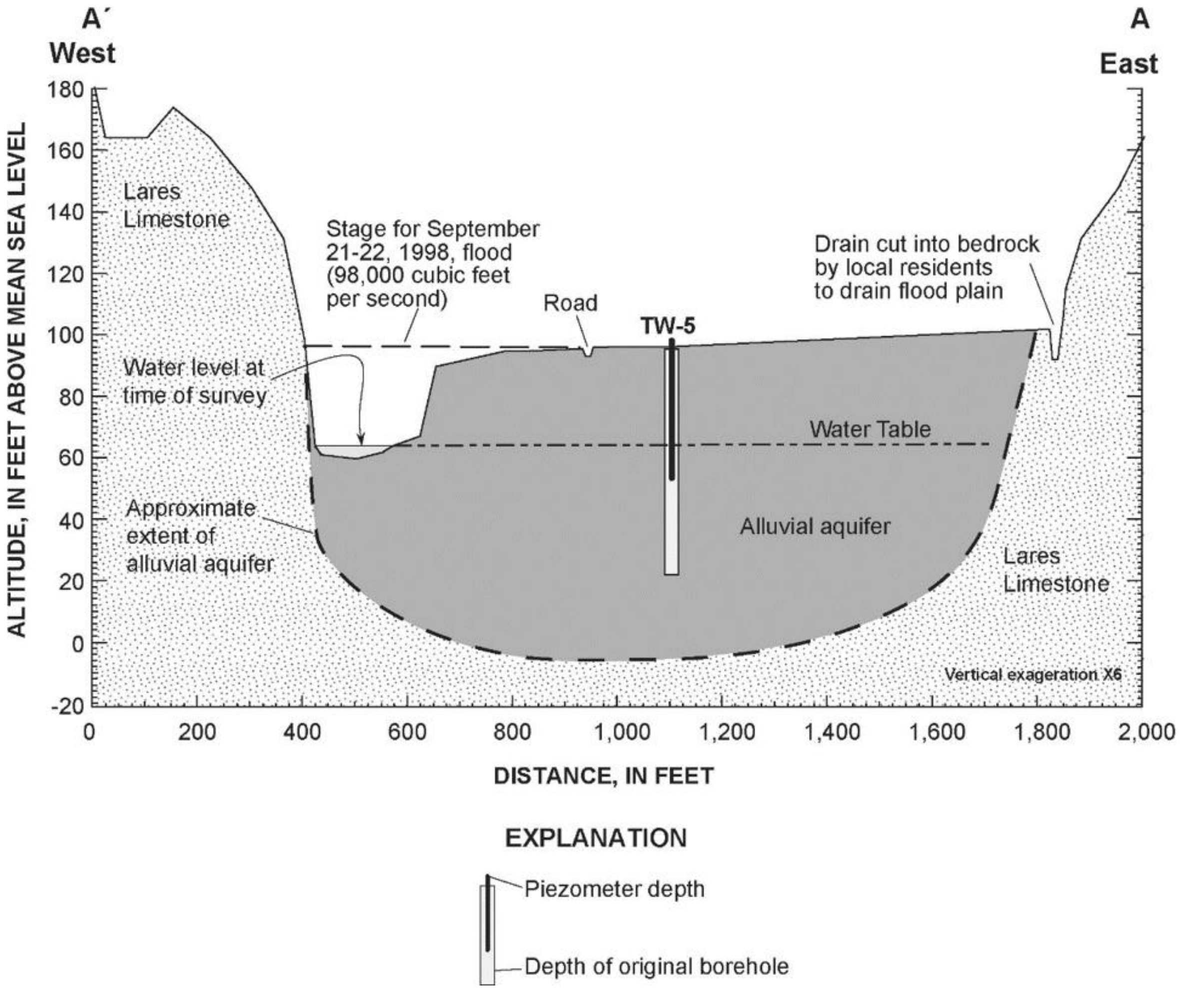
Geologic Setting

This is a small study area located on the north coast of Puerto Rico. Thick gravel floodplain alluvium is adjacent to the erosional channel cut by the Manati River (shown in yellow on the map below). The area of alluvium, where the city has permission to install a well, is proposed on the east side of the alluvium within the black rectangle.



Surface geologic map showing quaternary alluvial deposit to be used to filter river water (Torres-Gonzales and others, 2002).

Cross-section A-A′ is shown in the following image. The bedrock, which is limestone and dolomite, has low hydraulic conductivity. The gravels closest to the current river channel are coarsest and contain less fine material than the rest of the alluvium. Typically, gravel porosity ranges from 0.2 to 0.35 and hydraulic conductivity ranges from 1 to 40 m/day. The average land surface in the study area is 28 m above sea level (local vertical datum) and on average the base of the alluvium is about 0 m relative to the local sea level datum as shown in the cross section.

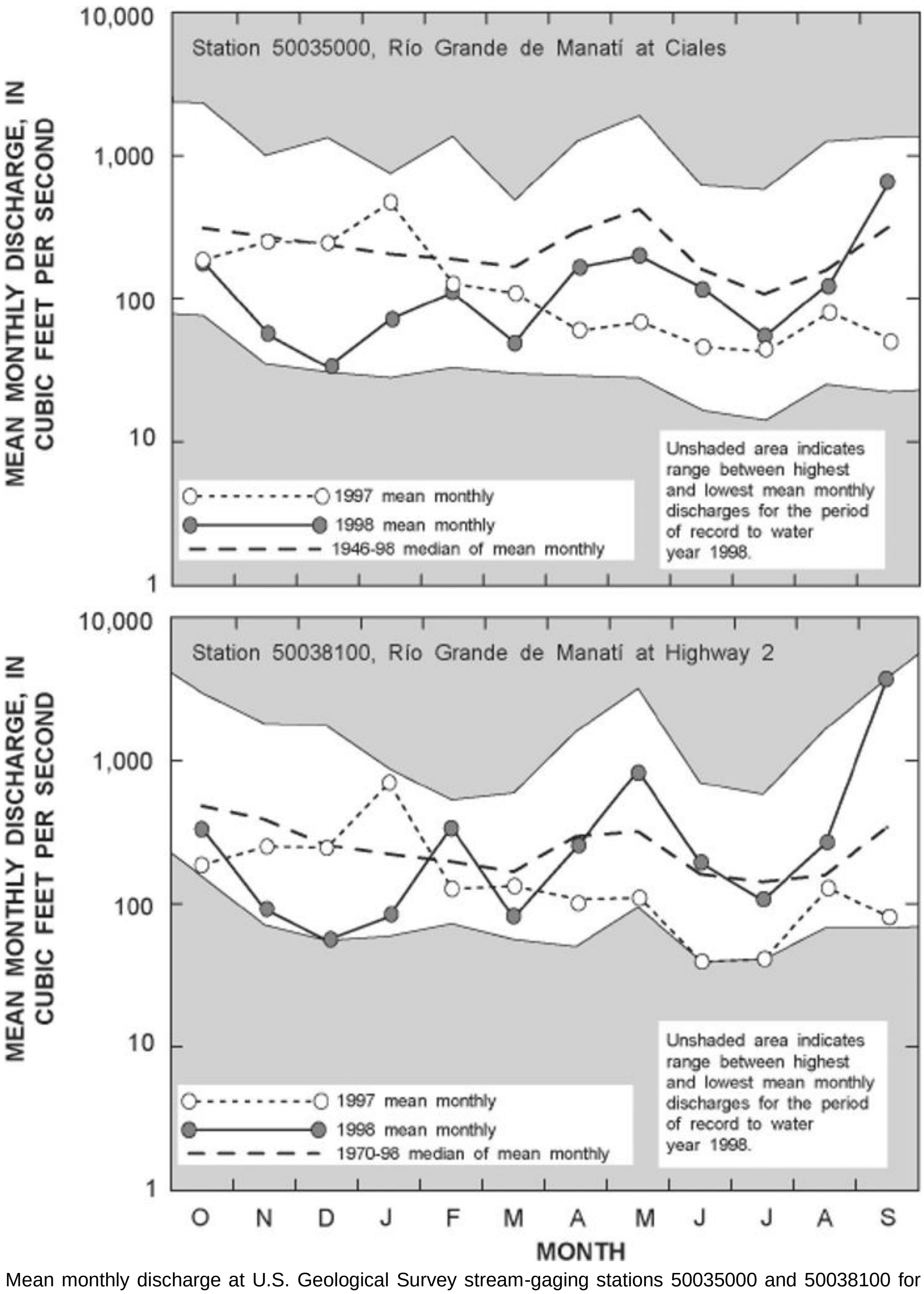


Cross section A-A' for Exercise 8. Units of length and altitude are in feet, showing average stage of river (Torres-Gonzales and others, 2002). 1 foot = 0.3048 meters.

Hydrology of Site

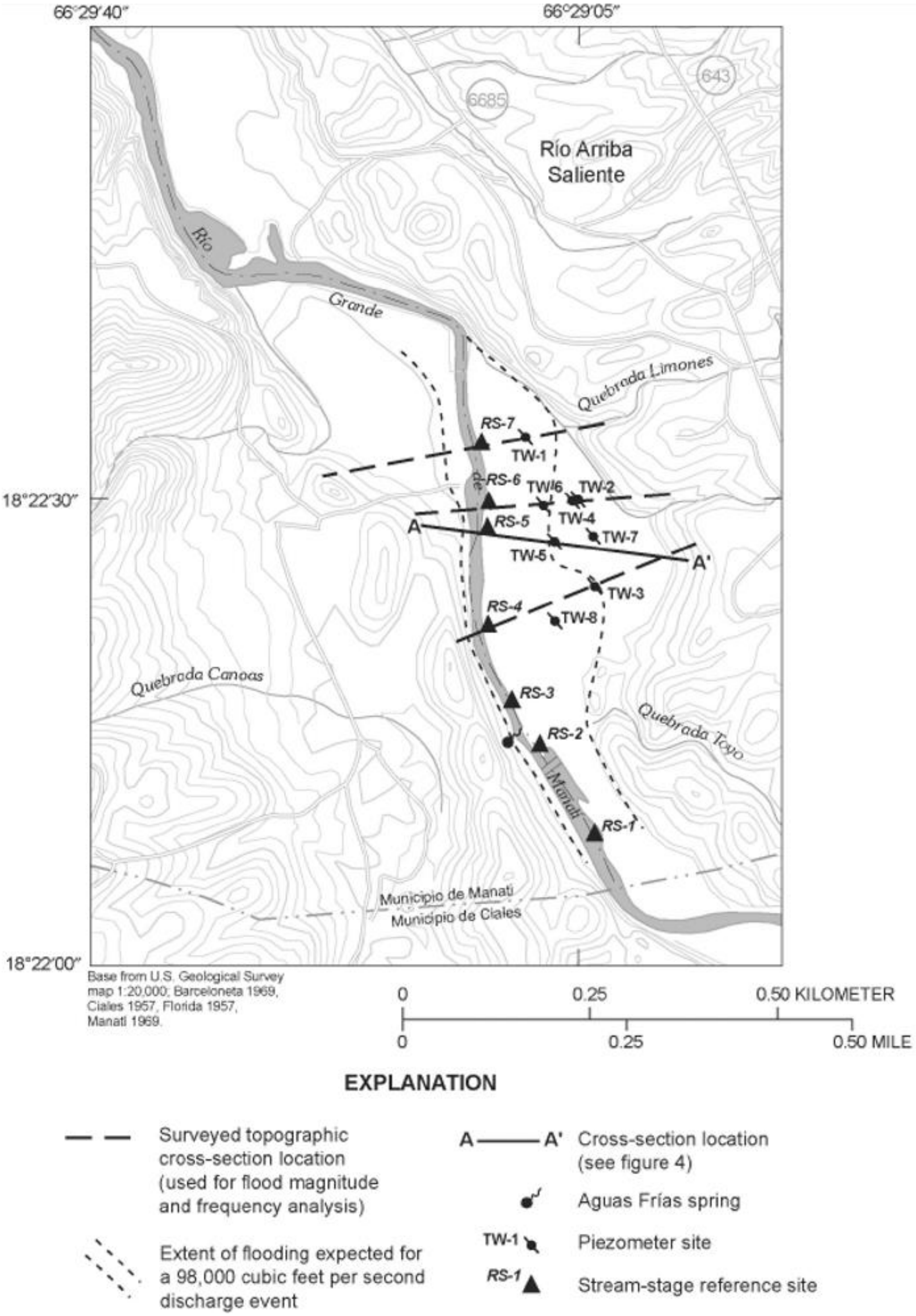
Puerto Rico has a humid tropical climate. The study area is approximately 18 degrees north of the equator. Thus, rainfall and temperature do not have seasonal signatures, with the exception of large rain events of short duration from hurricanes and tropical storms. Annual rainfall is typically 1549 mm and the estimated average net recharge per year is 154.9 mm or 0.0004245 m/day.

The river has fairly steady flow and stage except during the large storms as shown in the image below.



Mean monthly discharge at U.S. Geological Survey stream-gaging stations 50035000 and 50038100 for 1997 and 1998 water years (Torres-Gonzales and others, 2002). Also shown are the highest and lowest mean monthly discharge, as well as the long-term mean. The approximate locations of gaging stations 50035000 and 50038100 are shown in the map of the study area. Most of the 1997 and the first half of the 1998 water year were drier than normal. 1 cubic foot per second = 2446.6 cubic meters per day.

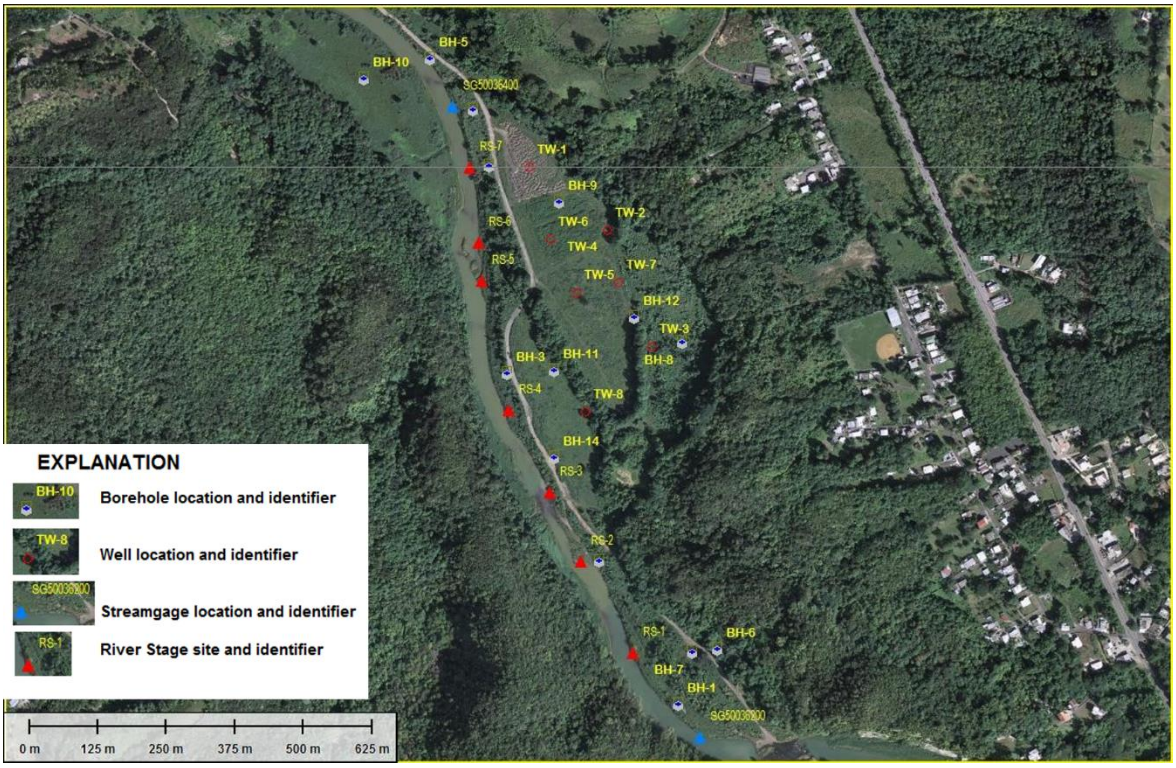
The extent of a significant flood event is shown in the image below indicating areas near the river that are periodically inundated during large storms.



Extent of significant flooding event, location of piezometers, and location of selected sampling sites within the Río Grande de Manatí valley, Río Arriba Saliente area, Puerto Rico (Torres-Gonzales et al., 2002).

Aerial Photograph of Data Locations

In April and May of 1998 exploratory data were collected using a cone penetrometer to establish depth to bedrock. The refusal depth indicates the bottom of the alluvial gravel (BH-# for borehole identifier), some of these points were completed with wells (TW-#, for well identifier). The image below shows the locations of data collection sites and the following table provides geographic coordinates in WGS84 and other site information for wells and boreholes. Eight wells were completed and slug tests at the wells indicate horizontal conductivity ranges from 14 to 32 m/day.



Location of data collection sites (image is testsites.jpg).

Data Locations and Values

Tables of data locations and values follow.

Data and information on sites where data were collected

Longitude (WGS84)	Latitude (WGS84)	Identi-fier	Site type	CPT refusal meters	Well depth meters	Case diam cm	Screen below surface meters	Altitude land surface meters	K_h in meter/day from slug tests
-66.485488	18.375000	TW-1	Well - Active	20	13	10	10 to 13	28.54	16
-66.484202	18.373961	TW-2	Well - Active	19	16	10	13 to 16	29.11	20
-66.483483	18.372057	TW-3	Well - Active	22	13	10	10 to 13	29.61	14
-66.484232	18.373941	TW-4	Well - Active	34	19	10	7 to 19	29.28	20
-66.484570	18.370998	TW-8	Well - Active	25	12	5	9 to 12	28.40	35
-66.484035	18.373109	TW-7	Well - Active	28	12	5	9 to 12	29.06	25
-66.485138	18.373811	TW-6	Well - Active	26	12	5	9 to 12	28.64	34
-66.484722	18.372940	TW-5	Well - Active	28	12	10	9 to 12	29.28	32
-66.483021	18.366195	BH-1	borehole	28					
-66.484321	18.368539	BH-2	borehole	29					
-66.485824	18.371597	BH-3	borehole	27					
-66.486130	18.375000	BH-4	borehole	30					
-66.487098	18.376744	BH-5	borehole	27					
-66.482384	18.367087	BH-6	borehole	22					
-66.482792	18.367036	BH-7	borehole	26					
-66.482970	18.372107	BH-8	borehole	23					
-66.484983	18.374400	BH-9	borehole	30					
-66.488169	18.376413	BH-10	borehole	28					
-66.485060	18.371648	BH-11	borehole	27					
-66.483751	18.372524	BH-12	borehole	28					
-66.486385	18.375903	BH-13	borehole	30					
-66.485060	18.370221	BH-14	borehole	31					
-66.483805	18.367053	RS-1	river stage site						
-66.484645	18.368557	RS-2	river stage site						
-66.485152	18.369677	RS-3	river stage site						
-66.485835	18.371024	RS-4	river stage site						
-66.486272	18.373141	RS-5	river stage site						
-66.486325	18.373770	RS-6	river stage site						
-66.486464	18.375000	RS-7	river stage site						
-66.486744	18.375992	SG50036400	Stream Gage						
-66.482693	18.365691	SG50036200	Stream Gage						

On 28 May of 1998, water level and river stage data were collected at the site. Shortly thereafter, a step-drawdown test was conducted. When interpreting the step drawdown test, it was assumed that the initial water levels had not changed. Data from the testing are shown in the following table.

Water Levels in Wells and at River Sites

Water levels in wells and at river sites

Identifier	Water level elevation immediately prior to test (meters above local sea level datum)
TW-1	19.33
TW-2	19.37
TW-3	19.38
TW-4	19.40
TW-5	19.38
TW-6	19.36
TW-7	19.42
TW-8	19.46
RS-2	20.12
RS-3	19.81
RS-4	19.46
RS-5	19.45
RS-6	19.16
RS-7	19.16
SG50036400	18.90
SG50036200	20.51

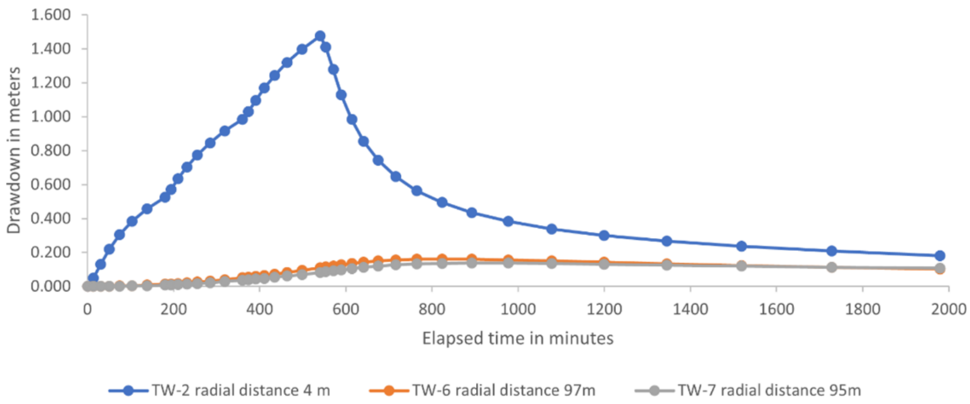
Step Test Pumping Rtaes for TW-4

For the step drawdown test, there were 3 pressure transducers and data loggers, which were used in wells TW-6, TW-7, and TW-2. TW-4 is close to TW-2 and is outside of the flood prone area. The step test was conducted to ensure that a 1,000 m^3/day withdrawal rate is possible at the site. The drilling company provided drawdown and elapsed time from beginning of pumping for those 3 wells as illustrated in the following graph.

Step test pumping rates for TW-4

Start time	Rate $meter^3$/day
8am	-1000
11am	-1500
2pm	-2000

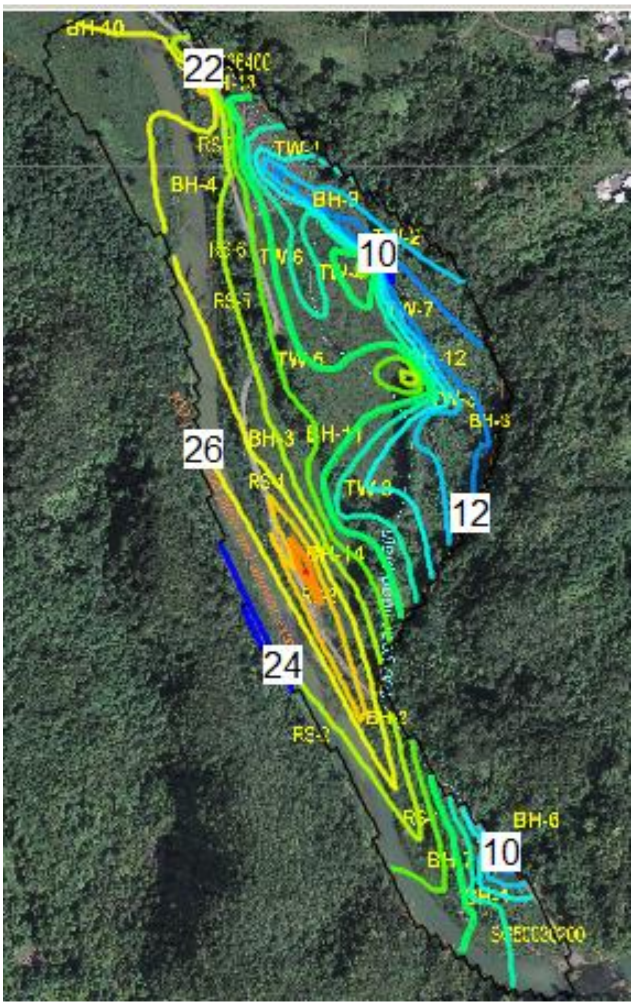
Drawdown Data at Observation Wells



Drawdown data at observation wells.

Saturated Thickness Map

The aquifer has a saturated thickness ranging from 10 to 32 m as shown in the following contour map.



Saturated thickness in the study area before pumping test (2-meter contour interval).

Step Test Analysis Challenges

Analysis of this test using an unconfined aquifer solution with partially penetrating wells yielded aquifer transmissivity and storage parameters. Unfortunately, the aquifer geometry and boundary conditions violate some of the assumptions of the analytical solutions (radial symmetry, constant thickness of infinite extent, and nearby no flow and recharge boundaries). In fact, none of the available analytical solutions are appropriate for this setting. The drilling company fit the drawdown at TW-2 to the Cooper-Jacob Straight line method (Cooper and Jacob, 1946) and then plotted the result using the Theis method to see if the fit was reasonable. From this analysis, the transmissivity is about 210 m^2/day and specific yield is about 0.03. If we assume the average saturated thickness of the aquifer is approximately 10 m in the test area (see map above), this results in a hydraulic conductivity of 20 m/day, which matches the slug test undertaken in TW-4.

Data Notes

The downloaded Digital Elevation Map (DEM) data are in the WGS84 datum (horizontal and vertical), the measured water level elevations are in the local Puerto Rico vertical datum and the tabulated latitude and longitude were estimated by eye for the WGS84 horizontal datum. Thus, for this Exercise, the DEM elevations are more than 5 meters too low, and the .xyz dataset from the ASTER GDEM v2 Worldwide Elevation Data was modified in the area of the alluvial aquifer to match the local Puerto Rico Datum. Additionally, the refusal depths were fabricated for this Exercise based on knowledge of the area in order to provide a realistic experience when developing the model. The step-test data were generated for this Exercise as were the slug test results.

[Solution to Exercise 8↓]

[Return to where text links to Exercise 8↑]

15 References

Anderson, M., W. Woessner, and R. Hunt, 2015, Applied Groundwater Modeling. Academic Press, 630 pages.

Cooper, H.H. and C.E. Jacob, 1946, A generalized graphical method for evaluating formation constants and summarizing well field history. Transactions, American Geophysical Union, volume 27, pages 526-534.

Dahl, T.E. and G.J. Allord, 1996, History of wetlands in the conterminous United States, in National water summary on wetland resources. United States Geological Survey Water-Supply Paper 2425, 8 pages, https://water.usgs.gov/nwsum/WSP2425/history.html↗.

Harbaugh, A.W., 2005, MODFLOW-2005, The United States Geological Survey modular ground-water model—the Ground-Water Flow Process. United States Geological Survey Techniques and Methods 6-A16, doi.org/10.3133/tm6A16↗.

Harbaugh, A.W., E.R. Banta, M.C. Hill, and M.G. McDonald, 2000, MODFLOW-2000, The United States Geological Survey Modular Ground-Water Model—User guide to modularization concepts and the Ground-Water Flow Process. United States Geological Survey Open-File Report 2000–92, 121 pages, doi.org/10.3133/ofr200092↗.

Harbaugh, A.W. and M.G. McDonald, 1996a, User's documentation for MODFLOW-96, an update to the United States Geological Survey modular finite-difference ground-water flow model. United States Geological Survey Open-File Report 96-485, 56 pages, doi.org/10.3133/ofr96485↗.

Harbaugh, A.W. and M.G. McDonald, 1996b, Programmer's documentation for MODFLOW-96, an update to the United States Geological Survey modular finite-difference ground-water flow model. United States Geological Survey Open-File Report 96-486, 220 pages, doi.org/10.3133/ofr96486↗.

Hill, M.C., E.R. Banta, A.W. Harbaugh, and E.R. Anderman, 2000, MODFLOW-2000, The United States Geological Survey modular ground-water model—User guide to the observation, sensitivity, and parameter-estimation processes and three post-processing programs. United States Geological Survey Open-File Report 00-184, 210 pages, doi.org/10.3133/ofr00184↗.

Hughes, J.D., C.D. Langevin, and E.R. Banta, 2017, Documentation for the MODFLOW-6 framework: United States Geological Survey Techniques and Methods, book 6, chapter A57, 40 pages, doi.org/10.3133/tm6A57↗.

Hughes, J.D., S.A. Leake, D.L. Galloway, and J.T. White, 2022, Documentation for the Skeletal Storage, Compaction, and Subsidence (CSUB) Package of MODFLOW 6: U.S. Geological Survey Techniques and Methods, book 6, chapter A62, 57 pages, doi.org/10.3133/tm6A62↗.

Kuniansky, E. L. and W.R. Danskin, 2003, Models Gone Bad—Common Modeling Problems and How to Solve Them, *In* MODFLOW and More 2003: Understanding through Modeling-Conference Proceedings edited by Poeter, Zheng, Hill, and Doherty: Colorado School of Mines, Golden, Colorado, 5 pages.

Kuniansky, E. L., C. J. Taylor, J. H. Williams, and F. Paillet, 2022, Introduction to Karst Aquifers. The Groundwater Project, Guelph, Ontario, Canada. doi.org/10.21083/978-1-77470-040-2↗.

Langevin, C.D., J.D. Hughes, E.R. Banta, R.G. Niswonger, S. Panday, and A.M. Provost, 2017, Documentation for the MODFLOW-6 groundwater flow model. United States Geological Survey Techniques and Methods, book 6, chapter A55, 197 pages, doi.org/10.3133/tm6A55↗.

Langevin, C.D., A.M. Provost, S. Panday, and J.D. Hughes, 2022, Documentation for the MODFLOW 6 Groundwater Transport Model. U.S. Geological Survey Techniques and Methods, book 6, chapter A61, 56 pages, doi.org/10.3133/tm6A61↗.

Lien, J.-M., L. Guilin, and C.D. Langevin, 2015, GRIDGEN version 1.0—A computer program for generating unstructured finite-volume grids. United States Geological Survey Open-File Report 2014–1109, 26 pages, doi.org10.3133/ofr20141109↗.

McDonald, M.G. and A.W. Harbaugh, 1984, A modular three-dimensional finite-difference ground-water flow model. United States Geological Survey Open-File Report 83-875, 528 pages, doi.org/10.3133/ofr83875↗.

McDonald, M.G. and A.W. Harbaugh, 1988, A modular three-dimensional finite-difference ground-water flow model. United States Geological Survey Techniques of Water Resources Investigations, book 6, chapter A1, 586 pages, https://pubs.usgs.gov/twri/twri6a1/↗.

Mehl, S.W. and M.C. Hill, 2006, MODFLOW-2005, The Unites States Geological Survey modular ground-water model—Documentation of shared node Local Grid Refinement (LGR) and the Boundary Flow and Head (BFH) package. United States Geological Survey, number 6-A12, doi.org/10.3133/tm6A12↗.

Mehl, S.W. and M.C. Hill, 2007, MODFLOW-2005, The US Geological Survey modular ground-water model—Documentation of the multiple-refined-areas capability of Local Grid Refinement (LGR) and the Boundary Flow and Head (BFH) package. United States Geological Survey Techniques and Methods 6-A21, 13 pages, https://pubs.usgs.gov/tm/2007/06A21/↗.

Mehl, S. and M.C. Hill, 2010, Grid-size dependence of Cauchy boundary conditions used to simulate stream-aquifer interactions. Advances in Water Resources, volume 33, number 4, 13 pages, doi.org/10.1016/j.advwatres.2010.01.008↗.

Niswonger, R.G., S. Panday, and M. Ibaraki, 2011, MODFLOW-NWT, A Newton formulation for MODFLOW-2005. United States Geological Survey Techniques and Methods 6-A37, 44 pages, doi.org/10.3133/tm6A37↗.

Panday, S., C.D. Langevin, R.G. Niswonger, M. Ibaraki, and J.D. Hughes, 2013, MODFLOW-USG version 1—An unstructured grid version of MODFLOW for simulating groundwater flow and tightly coupled processes using a control volume finite-difference formulation. United States Geological Survey Techniques and Methods, book 6, chapter A45, 66 pages, https://pubs.usgs.gov/tm/06/a45/↗.

Provost, A.M., C.D. Langevin, and J.D. Hughes, 2017, Documentation for the "XT3D" option in the Node Property Flow (NPF) Package of MODFLOW 6. United States Geological Survey Techniques and Methods, book 6, chapter A56, 40 pages, doi.org/10.3133/tm6A56↗.

Reilly, T.E., 2001, System and boundary conceptualization in ground-water flow simulation. United States Geological Survey Techniques of Water-Resources Investigations, book 3, chapter B8, 30 pages, http://pubs.usgs.gov/twri/twri-3_B8/↗.

Reilly, T.E. and A.W. Harbaugh, 2004, Guidelines for evaluating ground-water flow models. United States Geological Survey, Scientific Investigations Report 2004-5038, 30 pages, http://pubs.usgs.gov/sir/2004/5038/↗.

Shoemaker, W.B., E.L. Kuniansky, S. Birk, S. Bauer, and E.D. Swain, 2007, Documentation of a Conduit Flow Process (CFP) for MODFLOW-2005. United States Geological Survey Techniques and Methods, book 6, chapter A24, 50 pages, doi.org/10.3133/tm6A24↗.

Torres-Gonzales, S. and Gómez-Gómez, F., and Warne, A.G., 2002, Ground-Water Resource Assessment in the Rio Grande de Manati Alluvial Plain, Rio Arriba Saliente Area, Puerto Rico. U.S. Geological Survey Water-Resources Investigations Report, 02-4132, doi.org/10.3133/wri024132↗.

USGS MODFLOW 6 Development Team, 2022, MODFLOW 6 – Description of Input and Output, version mf6.3.0-March 4, 2022. United States Geological Survey website, https://water.usgs.gov/water-resources/software/MODFLOW-6/mf6io_6.3.0.pdf↗.

Winston, R.B., 2019, ModelMuse version 4—A graphical user interface for MODFLOW 6, United States Geological Survey Scientific Investigations Report 2019 – 5036, 10 pages, doi.org/10.3133/sir20195036↗.

16 Boxes

Box 1 Some Details of How MODFLOW 6 Works

MODFLOW simulates groundwater flow by solving the partial differential equation for groundwater flow using a block-centered finite volume method at the center of each active cell in the model. This forms a linear system of equations in the form of a matrix, [A], multiplied by the head at each cell in the form of a vector, H, set equal to flow into or out of each cell in the form of a vector, Q ([A]×H=Q). The equations represent the flow between the cell and its neighbors, the flow into or out of the groundwater system, and for transient simulations, the changes in the amount of stored water.

The components of the matrix [A] are computed from the specified model properties and the dimensions of each cell. MODFLOW solves for the groundwater level at the center of each cell (head vector) based on known flows into or out of specific cells in the form of well withdrawals or injections, known recharge to specific cells, and / or head-dependent fluxes specified at specific cells. MODFLOW solves the equations iteratively. On each iteration, it approximates the solution and then uses that approximate solution to help it find a better solution on the next iteration.

To use MODFLOW, one must have some knowledge of the properties of the aquifer, the dimensions of the aquifer and confining units, where water enters and exits the aquifer, and the quantity of water entering and leaving the aquifer. With MODFLOW, an initial head must be specified even if the initial head is unknown. Initial heads are described in more detail in Chapter 6.

At each time step, the iterative solver is used to find the unknown head at each cell by computing a new head value on each iteration within that time step until the head change and flow residual from one iteration to the next is smaller than thresholds set by the user. The head and flow thresholds are called the convergence criteria. For each succeeding time step, the heads from the previous converged time step will be used as the starting point for the next iteration. Using these heads, the solver calculates new values for the flows and heads for the current time step. For transient models there are multiple time steps and possibly multiple stress periods. A stress period is based on when the specified flows or head-dependent flows change. Stress periods can have multiple time steps or one time step. For a steady-state model, there is only one time step and stress period is needed because in such models, heads and flows do not change with time.

If the model is poorly constructed or there are numerical problems, MODFLOW may not be able to find an acceptable solution and will halt with an error message indicating the model did not converge. Each site model is different and may have different issues, such as rounding errors owing to the different properties of the aquifer and different specified flows. For this reason, there are multiple solver parameter options for the linear system of equations formed. Often getting the first time step to converge, using the initial conditions specified, for a transient or steady-state model can be difficult and may require adding time steps or modification of solver parameters. Even with a transient model, the first stress period may be a steady-state stress period so that the transient part of the simulation starts from initial heads that are consistent with the stresses on the system.

Kuniansky and others (2003) provide insights related to developing well-behaved models. The following offers an overarching perspective on the issues faced by modelers.

> *"From a mathematical perspective the groundwater flow equation is a second order linear or quasi linear equation of elliptic form and varies from the well-known Laplace equation when there are fluxes (wells or recharge) making the equation in the form of the Poisson equation. A partial differential equation in elliptic form generally forms matrices with nice numerical properties regardless of the applied method. When the equation is solved for a site problem set up with MODFLOW, a well-behaved system of equations is always formed if the grid is equally spaced, the aquifer is confined and homogeneous, and simple boundary conditions are applied. However, a poorly conditioned or unstable system of equations can sometimes occur, which results in slow convergence or lack of convergence of MODFLOW's iterative solvers. A poorly conditioned or unstable system of equations can be generated by inappropriate model discretization, extreme changes (several orders of magnitude) in hydraulic conductivity or transmissivity between aquifer zones, the specific implementation of piecewise linear head-dependent fluxes or inappropriate mixing of multiple head-dependent functions, thin model layers that convert between wet and dry, data entry mistakes, or incorrect conceptualization of the aquifer system."*

Return to where text links to Box 1↑

Box 2 Contents of the MODFLOW 6 Distribution Folder

Besides the MODFLOW program itself, the bin subdirectory of the MODFLOW 6 distribution folder contains two other programs: mf5to6.exe and zbud6.exe. mf5to6 is a program for converting MODFLOW-2005 model input files to MODFLOW 6 model input files. ZONEBUDGET (zbud6.exe) is used to extract the water budget of user defined zones within a MODFLOW 6 model.

The doc subdirectory contains documentation for MODFLOW 6. There are four reports documenting the theory of MODFLOW (tm6a55.pdf, tm6a56.pdf, tm6a57.pdf, and mf6suptechinfo.pdf). Another document (mf6io.pdf) describes the structure of the input and output files of MODFLOW 6. Release notes for the distribution are in release.pdf. The release notes include a description of the changes made to MODFLOW 6 along with other important information. The doc subdirectory also has documentation for mf5to6 and ZONEBUDGET along with a description of the example models.

The examples subdirectory contains the input files for a series of MODFLOW 6 models each in their own subdirectory. Each model subdirectory contains a file named "run.bat" You can run the model by double-clicking the run.bat file. The examples subdirectory contains a file named runall.bat which can be used to run all the example models.

The remaining subdirectories contain source code that can be used to compile MODFLOW 6, Mf5to6, and ZONEBUDGET.

Return to where text links to Box 2↥

Box 3 Test Your Knowledge of Structured Grids

Figure Box 3-1 is a plan view of a structured grid with five rows, five columns, and one layer. Which cells will have flow into or out of cell 13?

1	2	3	4	5
6	7	8	9	10
11	12	13	14	15
16	17	18	19	20
21	22	23	24	25

Figure Box 3-1 - Grid with numbered cells.

Answer: 8, 12, 14, and 18.

Cells 7, 9 17, and 19 are not adjacent because they do not share a cell face with cell 13. They have row and column numbers that both differ by one from the row and column number of cell 13. MODFLOW calculates flows between cell 13 and cells 8, 12, 14, and 18 but not between cell 13 and any other cell.

Return to where text links to Box 3↑

Box 4 Effect of Time-Step Size on Model Accuracy

Several MODFLOW-2005 simulations were executed using an example model to illustrate the effect of time-step size on accuracy. The model is a single layer model with 100 rows and 100 columns. The rows and columns are each 100 m wide and the layer has a uniform thickness of 100 m. The hydraulic conductivity is 0.0001 m/s and the specific storage is $1x10^{-5}$ m^{-1} thus the initial time step as estimated by Equation (2) is 250 s. A well with a pumping rate of -10 m^3/s is in row 50, column 50. The outline of the active cells is circular with a radius of 5000 m centered on the center of the grid and flow is simulated for 1,000,000 seconds (i.e., about 12 days). Simulations using 4, 10, 100, 1000, and 4000 constant-length (i.e., time-step multiplier = 1) time steps are executed. In two additional models, a time-step multiplier of 1.3 is used and the number of time steps is varied from 10 to 28.

Model results are plotted for two locations: one near the well, and one near the edge of the model as shown in Figure Box 4-1. The models are set up so that heads for every time step are saved during the first 100,000 seconds, and after that, heads are only saved at intervals of 100,000 seconds. The size of the initial time step as estimated by Equation (2) requires 4000, constant-length, time steps. However, using 1000 time-steps of four times the estimated length produces virtually identical results as the case with 4000 time-steps so the case with 4000 time-steps is not included in this presentation.

Thus, the most accurate solution is illustrated by the case with 1000 time-steps which is represented by the solid black line in Figure Box 4-1. At both monitored locations, an increase of time-step size (i.e., a decrease in the number of constant-length steps) results in less accuracy (For "100 Steps," the accuracy is very close to that of "1000 Steps" but note that at early time the head change near the well deviates slightly from the "1000 Steps" solution). For the models in which time-step multipliers were used; the accuracy was better than with models that used a similar number of steps without multipliers, as illustrated in Figure Box 4-1 where the results for "10 Steps Mult" are closer to those for "1000 Steps" than the results for "10 Steps." At the observation location close to the well, the greatest inaccuracies were in the initial time steps whereas at the location close to the model edge, the inaccuracy increased with time from 100,000 seconds to 300,000 seconds while the rate of head decline was increasing. After 300,000 seconds, the rate of head decline is more linear and the accuracy improved.

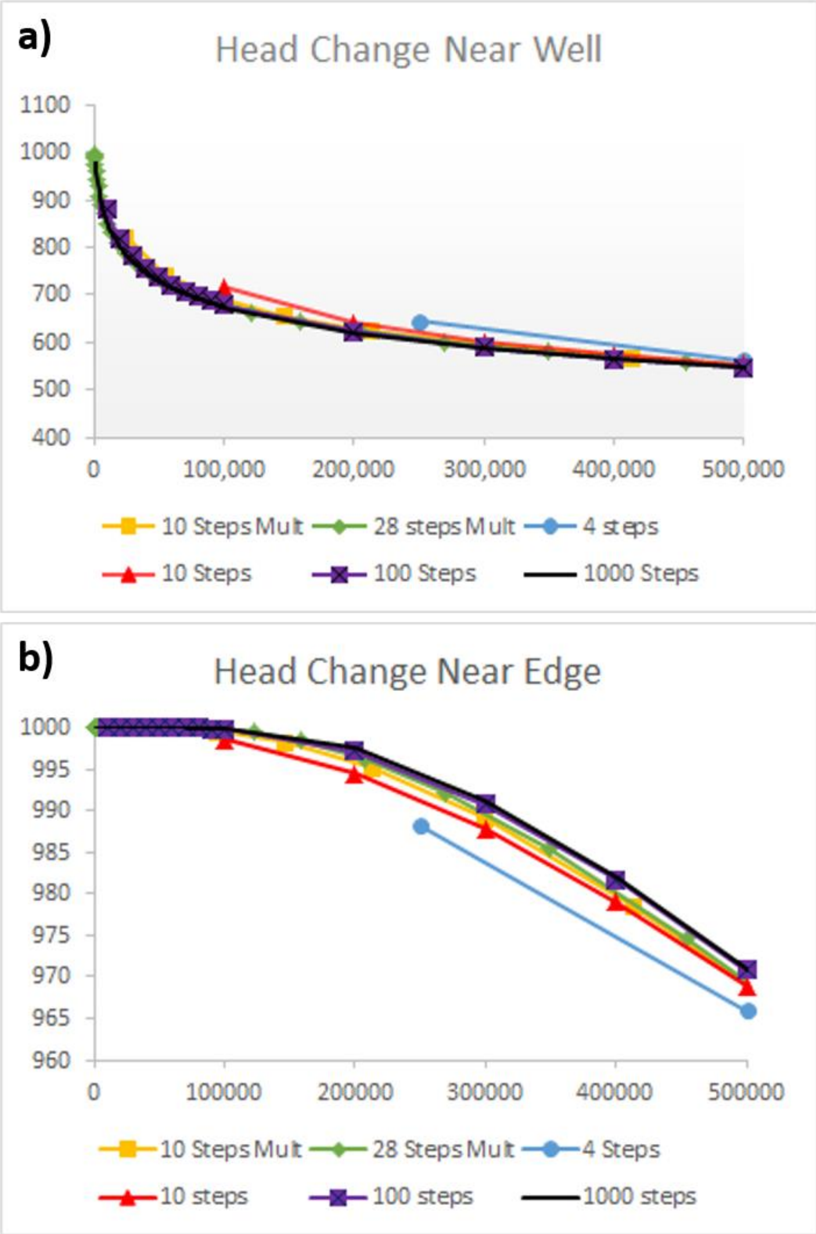


Figure Box 4-1 - Effect of time step size on calculated head. a) Observation location at row 50, column 48. b) Observation location at row 50, column 5.

Return to where text links to Box 4 ⮥

Box 5 Step-by-Step Instructions, Model PS1A

1. Start ModelMuse, create a new MODFLOW model and click *Next*. You can specify "not available" (N/A) as the projection for this example model. Specify the length and time units (meters and days). You can also specify a description of the model. Then click *Next* again (Figure Box 5-1).

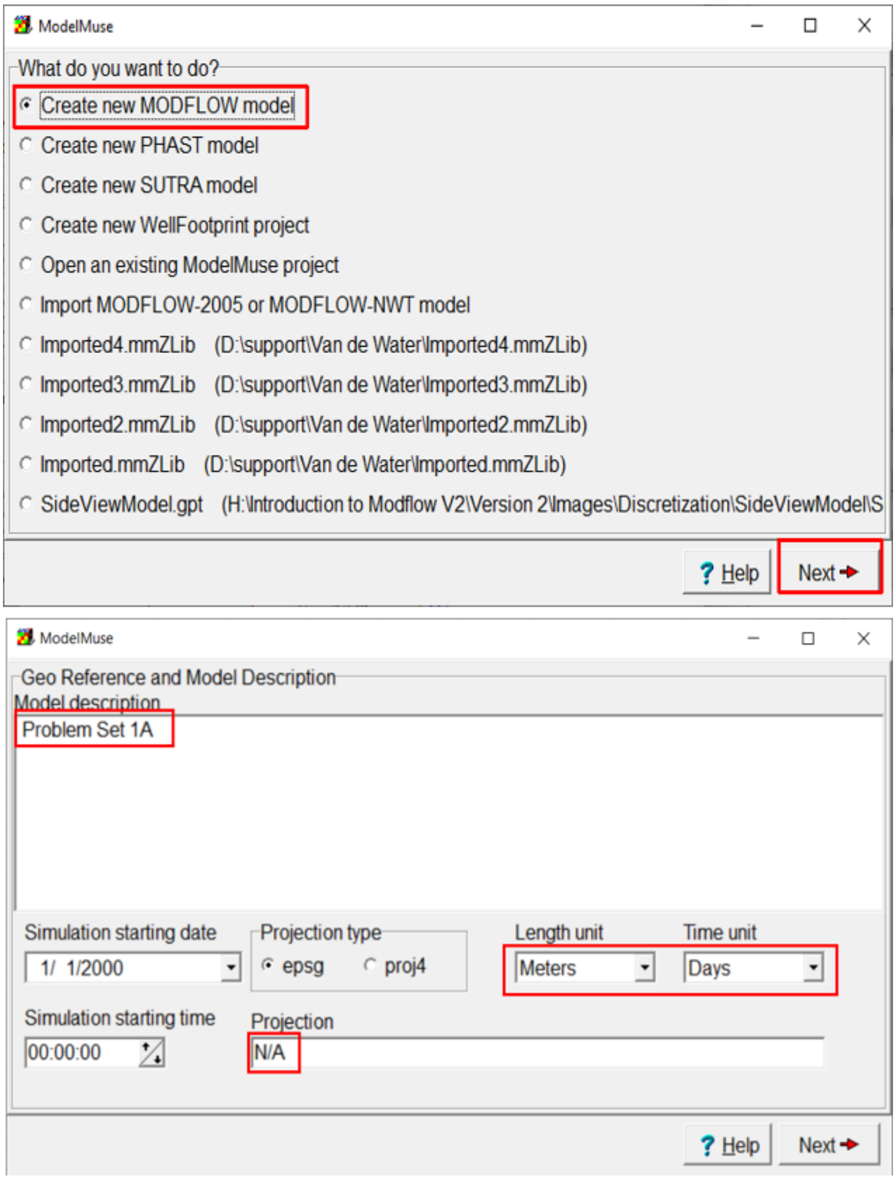


Figure Box 5-1 - Screen captures illustrating step 1.

2. Choose a MODFLOW 6 model and specify the grid. Click *Finish*. Then save your model as "PS1A" (Figure Box 5-2).

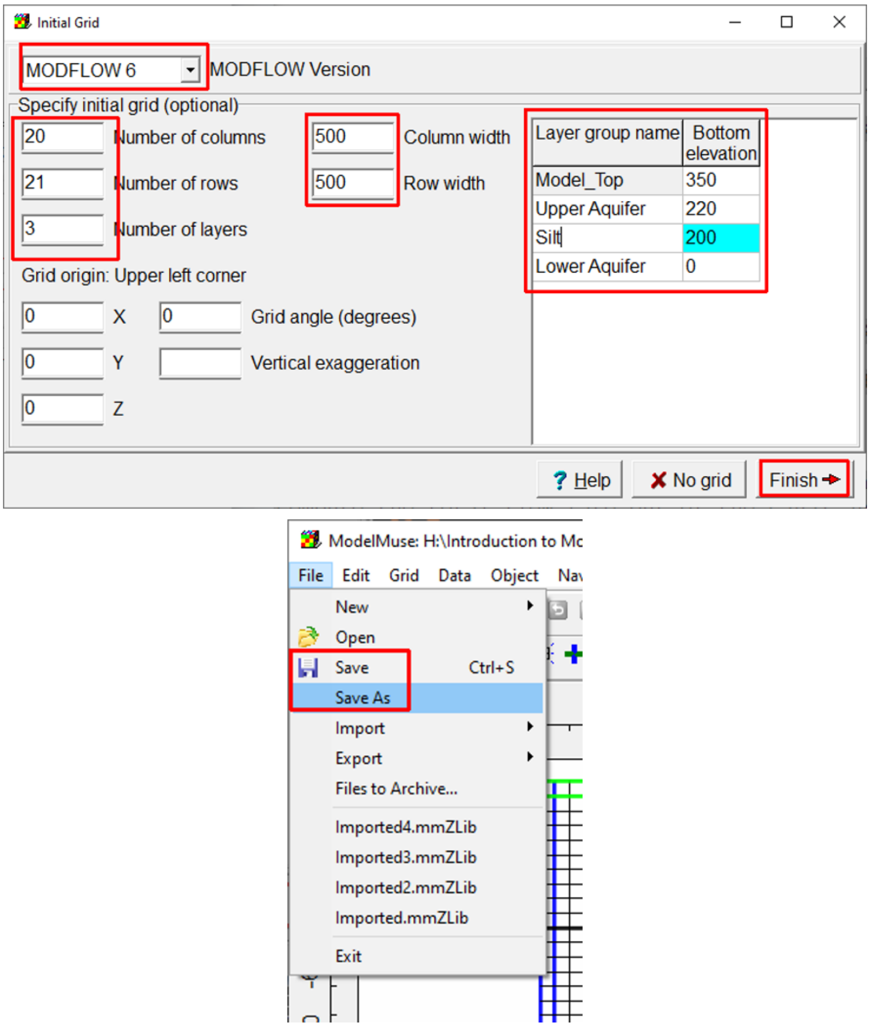


Figure Box 5-2 - Screen captures illustrating step 2.

3. Select *Data|Edit Data sets* and specify the initial head (Figure Box 5-3).

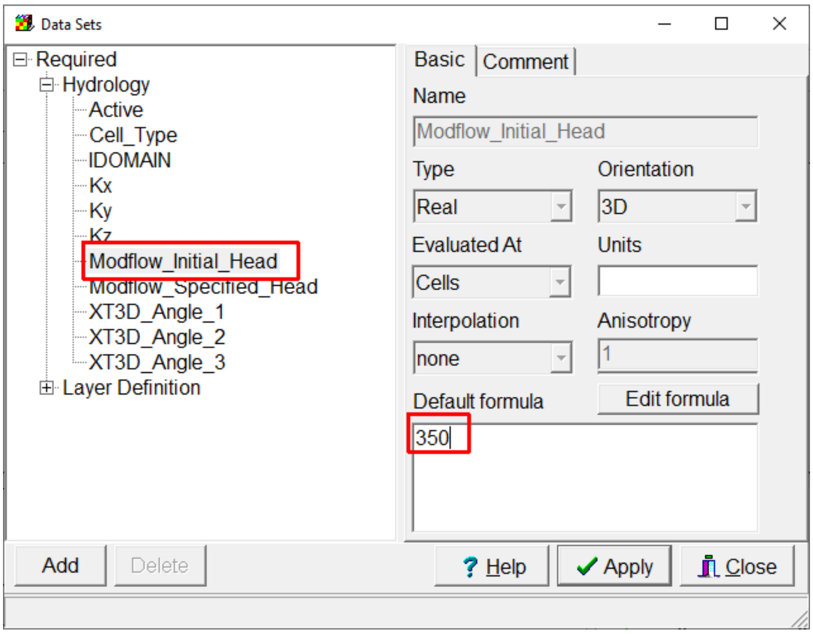


Figure Box 5-3 – Screen capture illustrating specifying the initial head.

4. Use a formula to make all the cells in the top layer convertible and the rest confined. The Cell_Type data set is used to make a cell confined or convertible. The "associated model data" of the Cell_Type data set, under the Comment tab, indicates that a value of one means that the cell's saturated thickness will be calculated based on the head whereas a value of zero means that the saturated thickness depends only on the top and bottom elevation of the cell. The top layer is layer 1, so the formula "`If((Layer=1),1,0)`" will do what we want. You can click the *Edit Formula* button to display the Formula Editor to help you create the formula, then click *OK*, *Apply* (Figure Box 5-4).

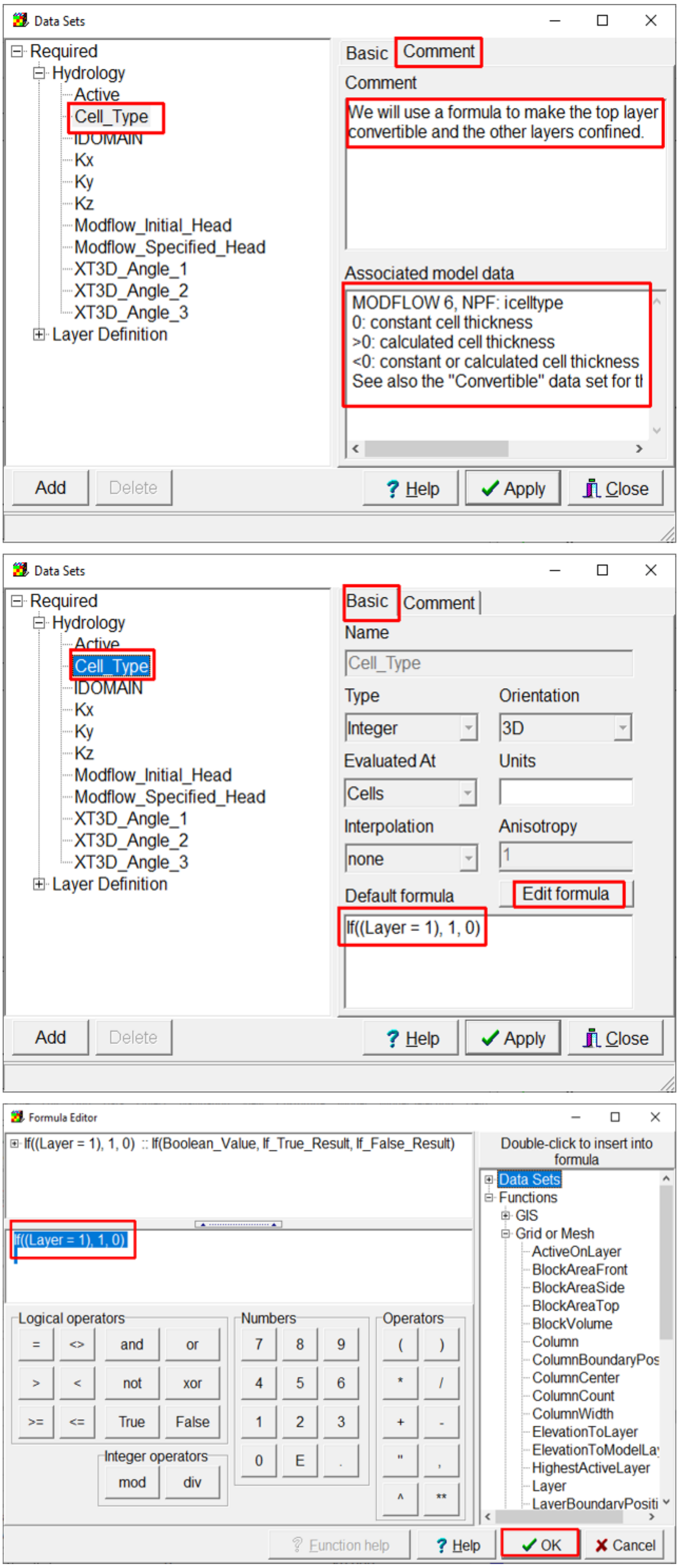


Figure Box 5-4 - Screen captures illustrating specification of the top layer as convertible.

5. Use a formula to specify Kx for each layer (Figure Box 5-5). Look up the Case function in the ModelMuse help to understand why this formula works. After using the formula editor, click *OK*, *Apply*.

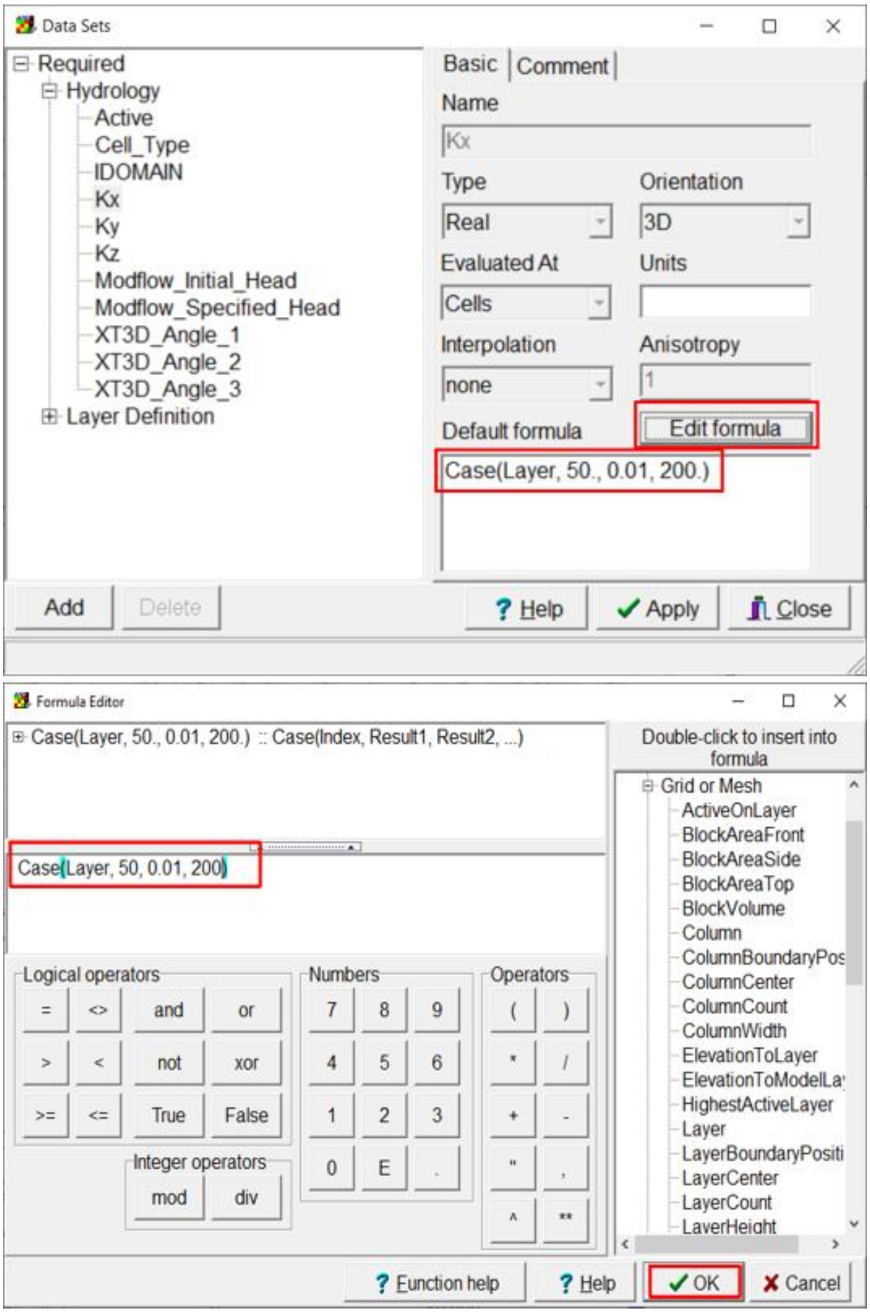


Figure Box 5-5 - Screen captures illustrating specifying the horizontal hydraulic conductivity.

6. Use a formula to specify Kz for each layer (Figure Box 5-6).

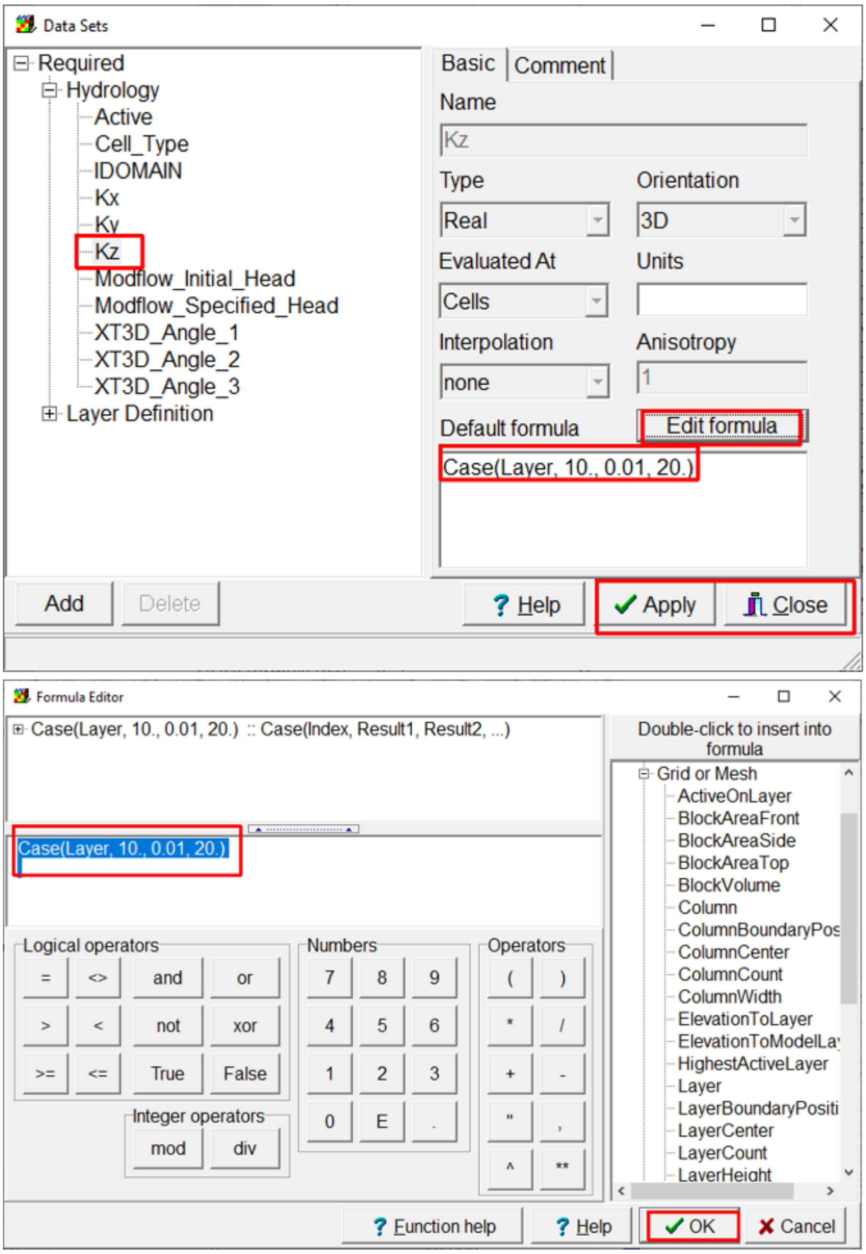


Figure Box 5-6 - Screen captures illustrating specifying vertical hydraulic conductivity.

7. Activate the CHD package. Select *Model|MODFLOW Packages and Programs* and check the check box for the CHD package (Figure Box 5-7). When a package is activated, its input file will be created the next time you generate the MODFLOW input files and the input file for the package will be listed in the MODFLOW name file.

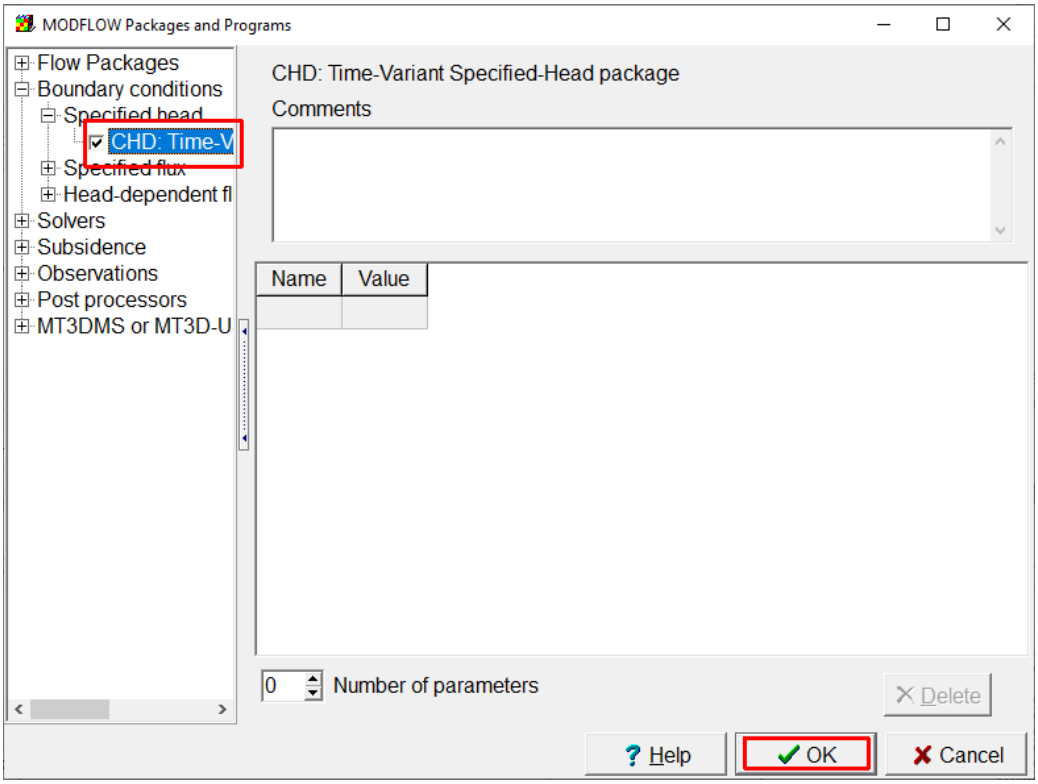


Figure Box 5-7 - Screen capture illustrating activating the CHD package.

8. Optional: MODFLOW iterates to arrive at a solution and the iteration process may stop before reaching a sufficiently accurate solution based on the specified closure criteria. Sometimes, this can be evident from a large percent discrepancy in the water budget of the model. This step will require a smaller head change between iterations before the iteration process stops. Decrease the nonlinear and linear head closure criteria in the Solver by selecting the *Model|MODFLOW Packages and Programs* option, expanding Solvers, and visiting first the nonlinear tab and then the linear tab, and while in each tab check the box to override the default DVClose and enter the values shown in Figure Box 5-8.

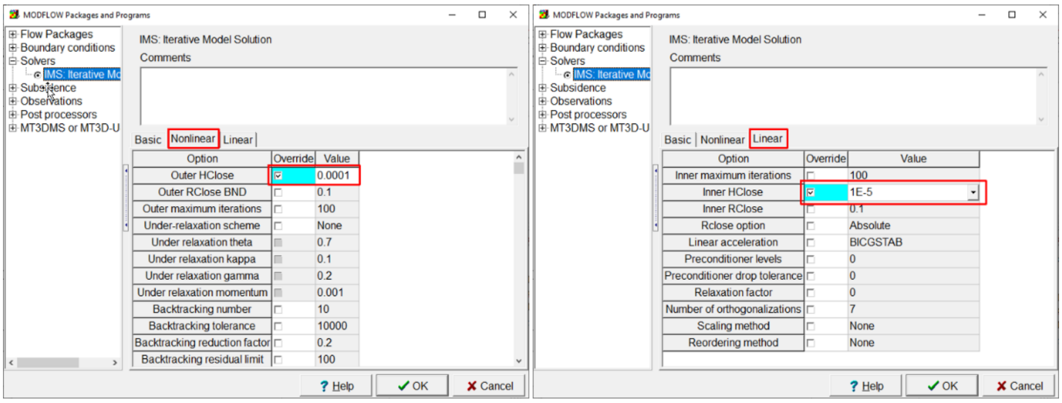


Figure Box 5-8 - Screen captures illustrating specification of the outer and inner head closure criteria.

9. Create a line object in the last column of the model and use it to specify a constant head in the top layer. To create a line object, select *Object|Create|Straight line* or select *Object|Create|Polyline*. Then click at one end of the line and then double-click at the other end of the line (Figure Box 5-9). When the Object Properties dialog box appears, use it to specify the position and stage of the river on the properties and MODFLOW features tabs (Figure Box 5-10).

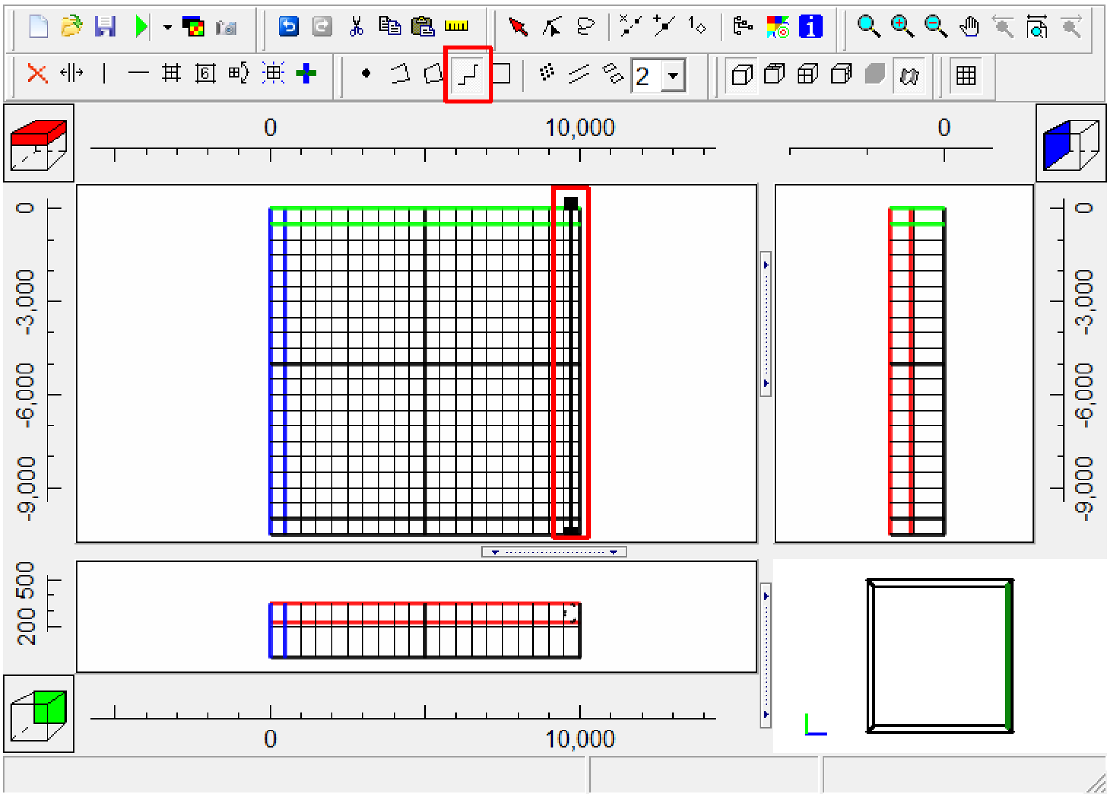
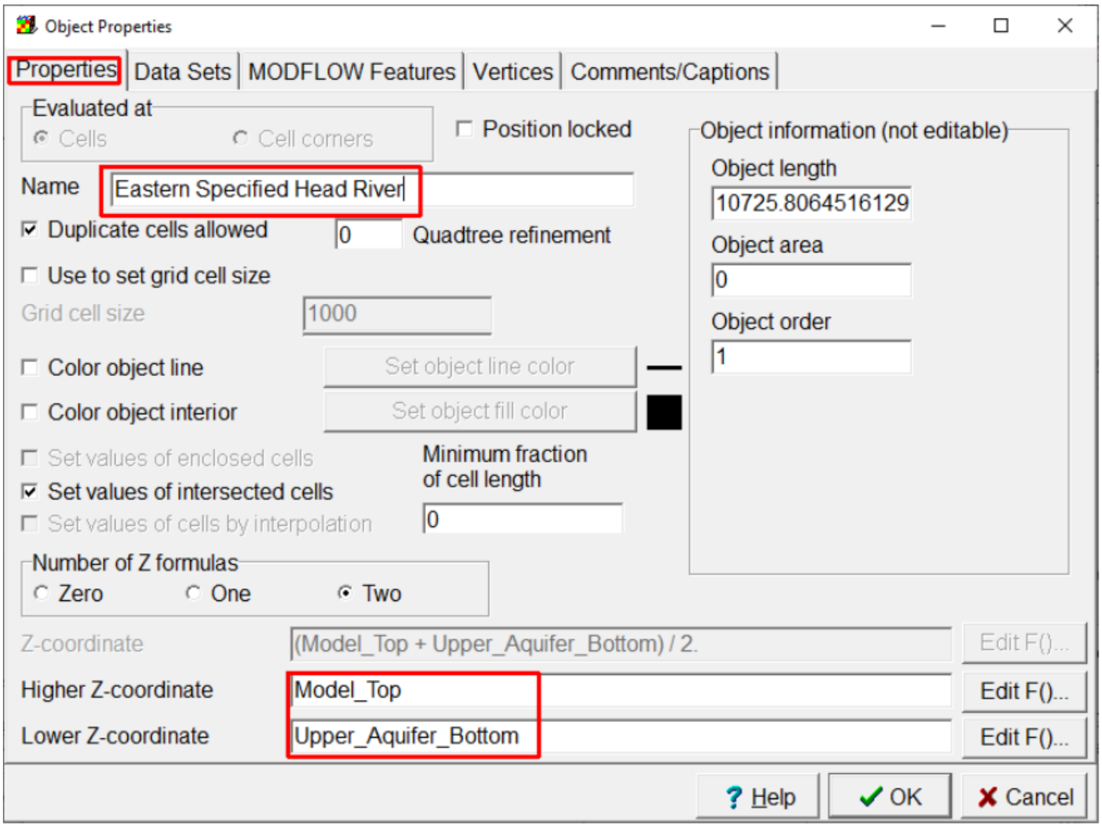


Figure Box 5-9 - Screen capture illustrating creating a straight-line object.



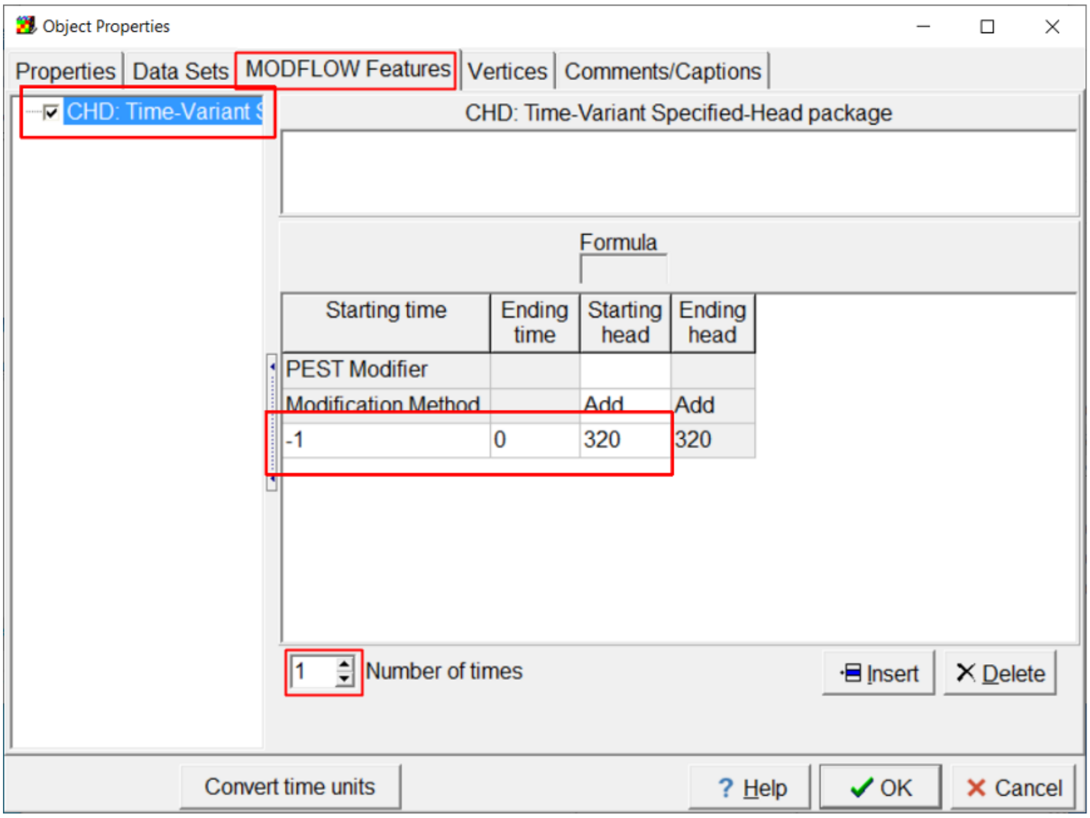


Figure Box 5-10 - Screen captures illustrating specifying a specified head boundary.

10. Create another line object in the first column and use it to specify a constant head in the top layer representing the canal (Figure Box 5-11).

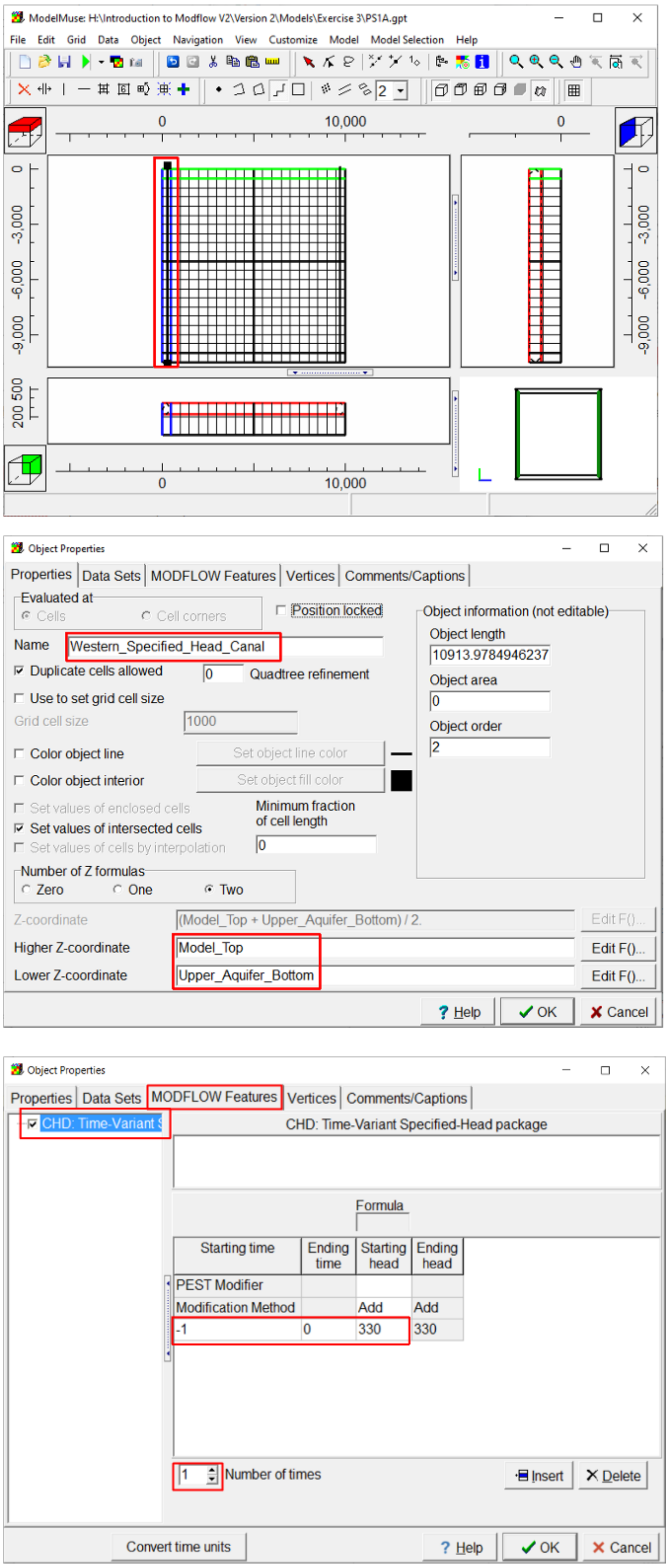


Figure Box 5-11 - Screen captures illustrating creating another specified head boundary.

11. Select *Model|MODFLOW Output Control* and specify how the data from the model should be printed (Figure Box 5-12).

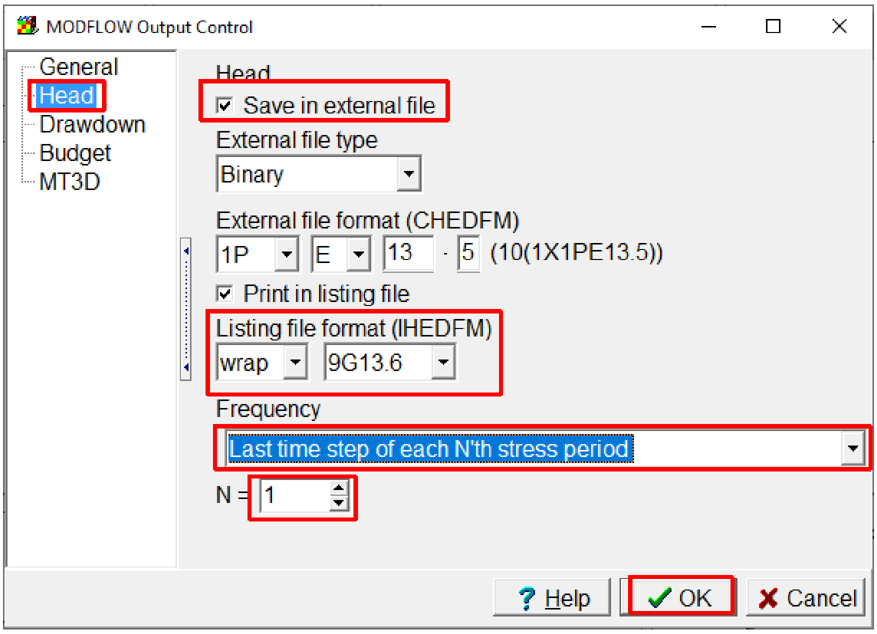


Figure Box 5-12 - Screen capture illustrating specifying how data from the model should be printed or saved.

12. Select *Model|MODFLOW Program Locations*. Make sure that the location of MODFLOW 6 is specified correctly. You may also wish to choose ListingAnalyst as the text editor used to open the MODFLOW listing file (Figure Box 5-13).

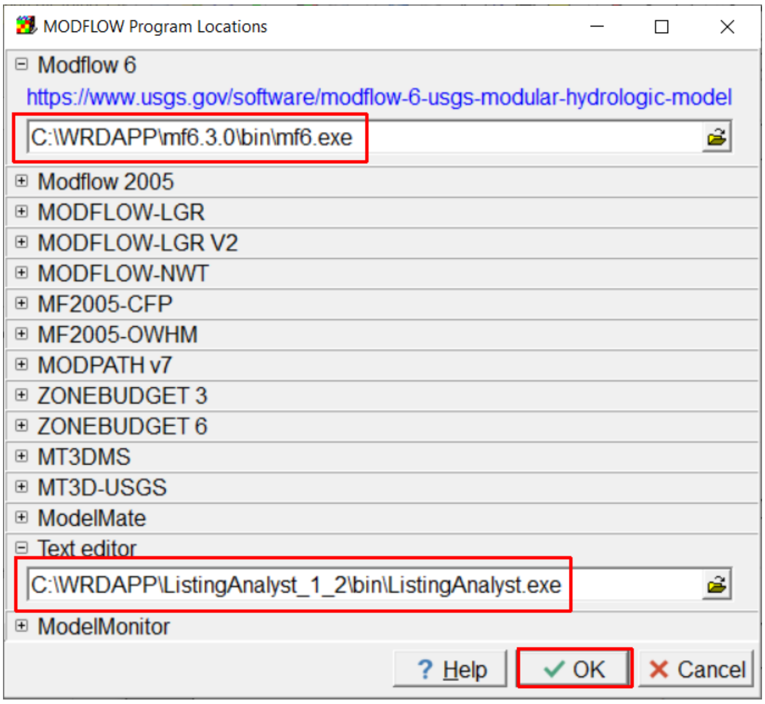


Figure Box 5-13 - Screen capture illustrating specifying executables to use with ModelMuse.

13. Select *File|Export|MODFLOW 6 input files* or click the *Run MODFLOW* button to run the model (Figure Box 5-14). Select *File|Save* to save PS1A. This version of the model will be modified and used again in several subsequent exercises.

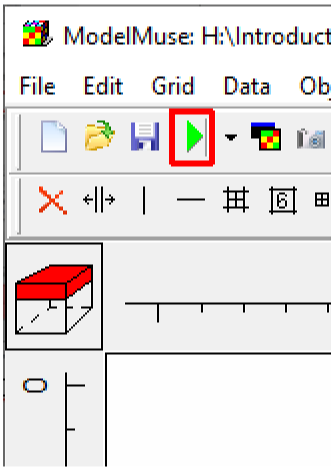


Figure Box 5-14 – Screen capture illustrating running MODFLOW from ModelMuse.

14. Close the ModelMonitor. This will prompt ModelMuse to open the listing file in a text editor. Use the information in the listing file to answer the questions posed in Exercise 3.1 through 3.4. In the illustration below the listing file is displayed in ListingAnalyst. ListingAnalyst allows you to quickly navigate in a MODFLOW listing file (Figure Box 5-15).

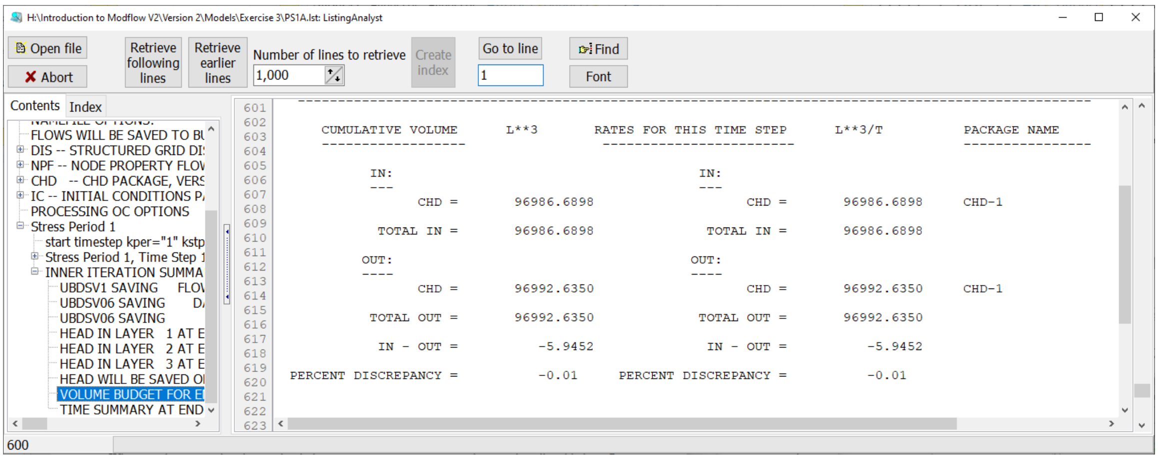


Figure Box 5-15 - Screen capture illustrating Listing Analyst displaying a MODFLOW output file.

Return to where text links to Box 5 ↥

Box 6 Step-by-Step Instructions for using ZONEBUDGET with Model PS1A

1. With PS1A open, save the model with a different name such as PS1A_ZB.
2. Select *Model|MODFLOW Packages and Programs* and activate ZONEBUDGET (Figure Box 6-1). Click *OK* to close the dialog box. A new integer data set will be created named "Zones" (Figure Box 6-2).

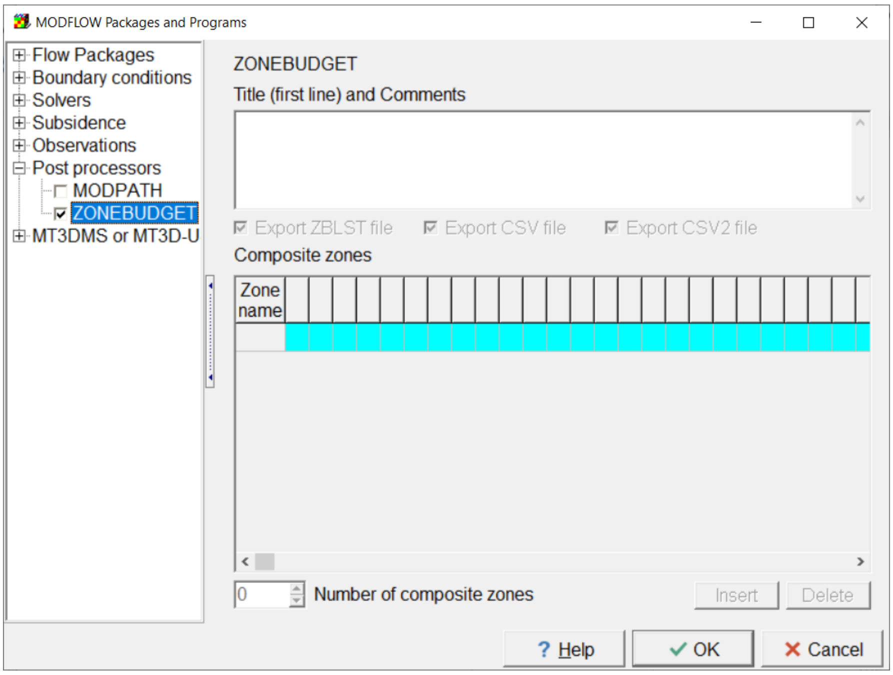


Figure Box 6-1 - Screen capture of the MODFLOW Packages and Programs dialog box illustrating the activation of ZONEBUDGET.

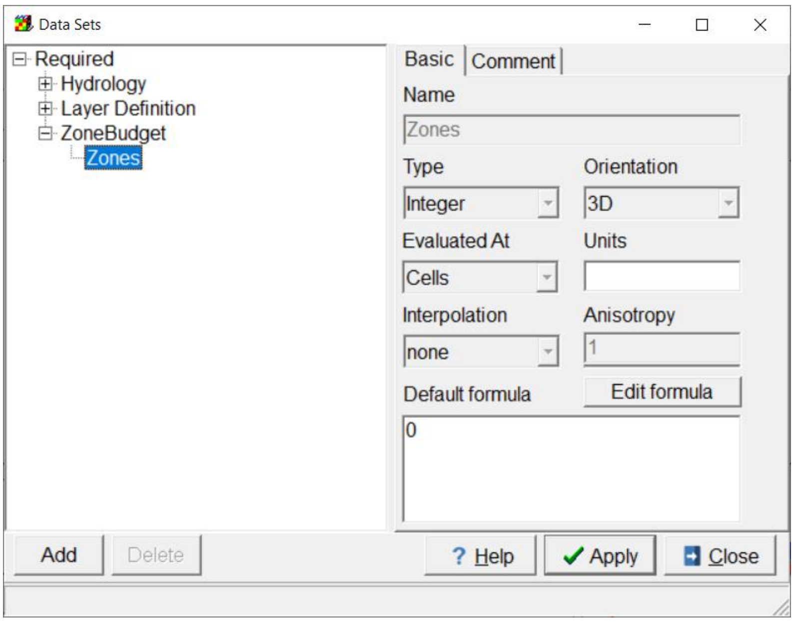


Figure Box 6-2 - Screen capture of the Data Sets dialog box illustrating the new data set "Zones."

3. Set the selected layer to 2. You can do this by selecting *Grid|Set Selected Col, Row, Layer…* or *Grid|Set Selected Column, Row, Layer*. The first command displays a dialog box to specify them. The second, allows you to click on the grid to select them. You can also change the selected layer by clicking on the red cube in the upper left corner of the top view of the model.

4. Select *Object|Create|Rectangle* and draw a rectangle object that covers the left half of the model grid. Use it to assign a zone number of 1 to the Zones data set in left half of the model in layer 2 (Figure Box 6-3).

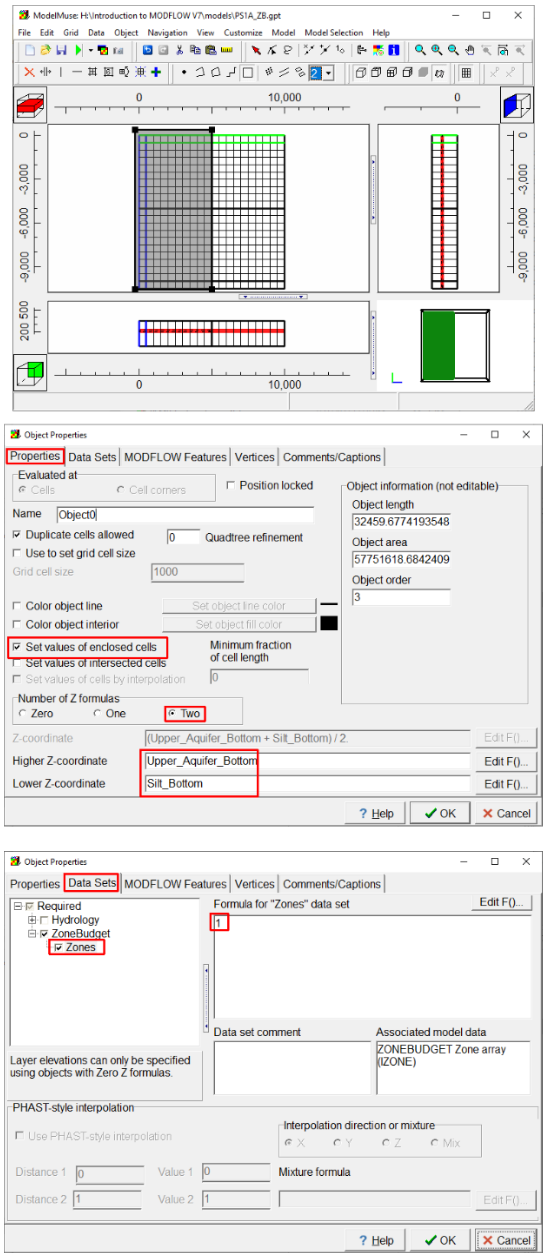


Figure Box 6-3 - Screen capture illustrating the specification of ZONEBUDGET zone 1.

5. Create additional rectangle objects to assign values of 2, 3, and 4, to the right half of the grid in layer 2, the left half of the grid in layer 3 and the right half of the grid in layer 3 respectively.

6. Color the grid with the Zones data set to check that the zones have been assigned correctly (Figure Box 6-4).

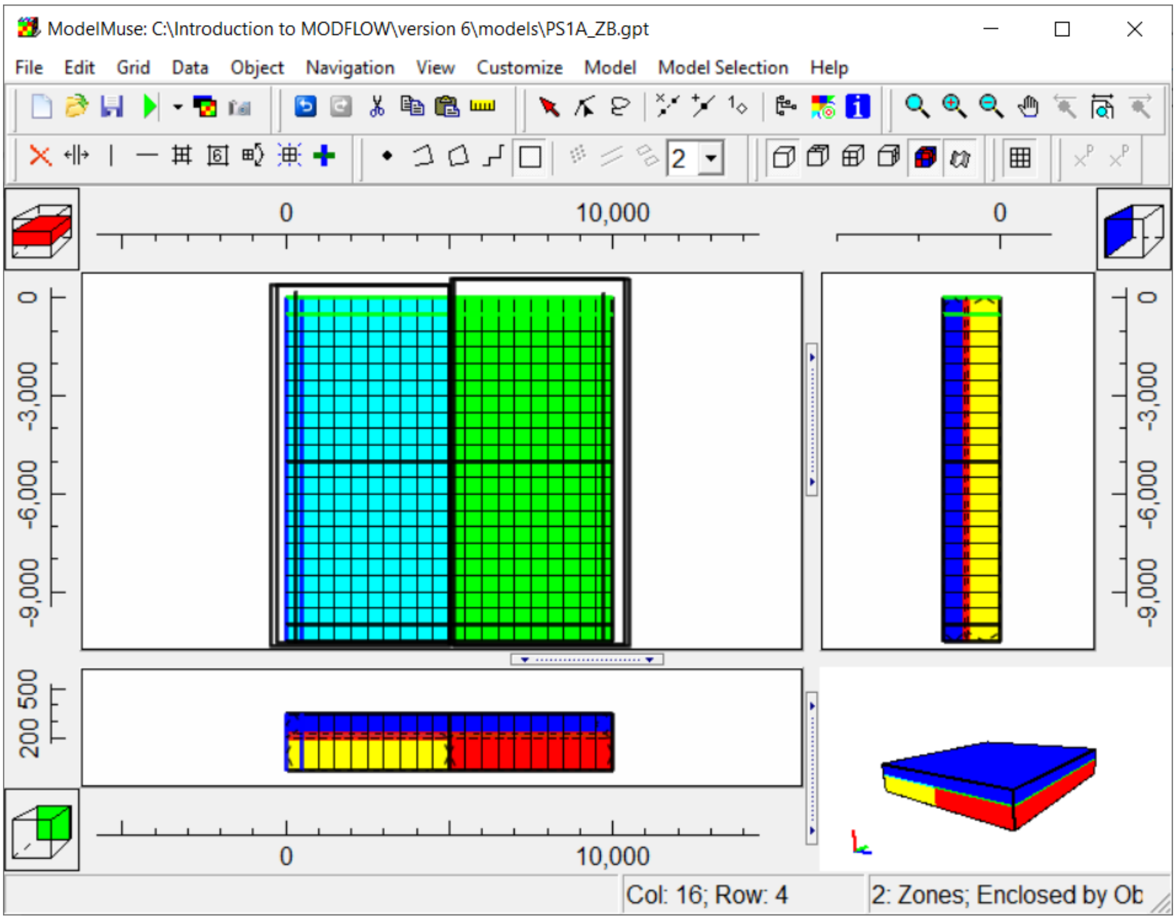


Figure Box 6-4 - Screen capture of ModelMuse main window illustrating the distribution of ZONEBUDGET zones.

7. ZONEBUDGET uses output files from MODFLOW as part of its input. Because the name of the model has been changed, you will have to run MODFLOW again before running ZONEBUDGET. Select *File|Export|MODFLOW 6 Input Files*. The Save MODFLOW input files dialog box has a check box for exporting the ZONEBUDGET input files. If that checkbox is checked, ZONEBUDGET will run immediately after MODFLOW 6. If you had already run the model, you could have selected *File|Export|ZONEBUDGET Input Files* to run ZONEBUDGET.

8. After running MODFLOW and ZONEBUDGET, open the ZONEBUDGET listing file (PS1A_ZB.zb.lst) in a text editor. You can determine the flows between zones from the budget for each zone (Figure Box 6-5).

```
 VOLUME BUDGET FOR ZONE 3 AT END OF TIME STEP    1, STRESS PERIOD   1
 ---------------------------------------------------------------------------------------------------

    CUMULATIVE VOLUME      L**3       RATES FOR THIS TIME STEP     L**3/T          PACKAGE/MODEL
    ------------------                 ------------------------                   ----------------

          IN:                                   IN:
          ---                                   ---
         DATA-SPDIS =          0.0000          DATA-SPDIS =          0.0000     NPF
                CHD =          0.0000                 CHD =          0.0000     CHD-1
             ZONE 1 =      53505.2120              ZONE 1 =      53505.2120
             ZONE 4 =          0.0000              ZONE 4 =          0.0000

           TOTAL IN =      53505.2120            TOTAL IN =      53505.2120

         OUT:                                  OUT:
         ----                                  ----
         DATA-SPDIS =          0.0000          DATA-SPDIS =          0.0000     NPF
                CHD =          0.0000                 CHD =          0.0000     CHD-1
             ZONE 1 =          0.0000              ZONE 1 =          0.0000
             ZONE 4 =      53505.1937              ZONE 4 =      53505.1937

          TOTAL OUT =      53505.1937             TOTAL OUT =      53505.1937

           IN - OUT =      1.8280E-02              IN - OUT =      1.8280E-02

PERCENT DISCREPANCY =          0.00     PERCENT DISCREPANCY =          0.00
```

Figure Box 6-5 - Screen capture of the ZONEBUGET listing file with the flow from zone 3 to zone 4 highlighted. This is the flow from layer 2 to layer 3 in the left half of the model.

Return to where text links to Box 6↑

Box 7 Step-by-Step Instructions for Model PS1B1

1. Open PS1A and save as PS1B1.
2. Modify the description of the project in the *Model|MODFLOW Options* dialog box (Figure Box 7-1).

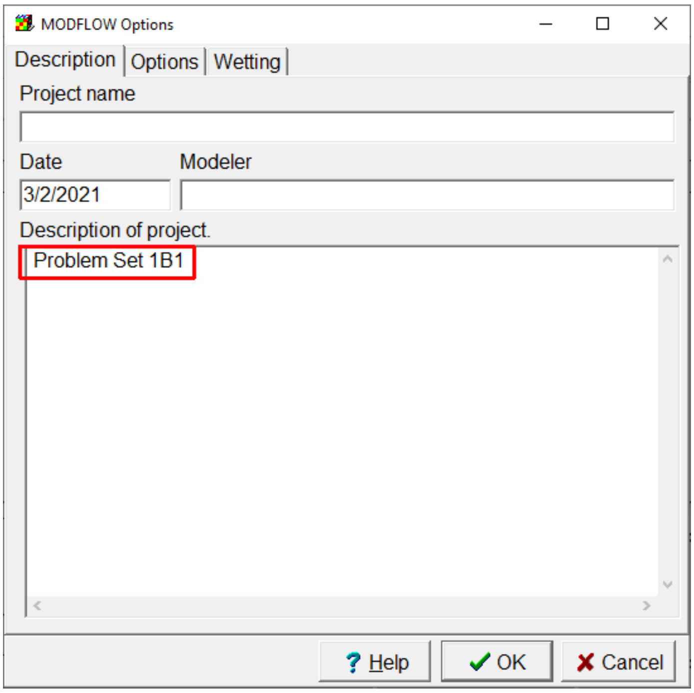


Figure Box 7-1 - Screen capture illustrating modification of the project description.

3. In the Edit Data Sets dialog box modify the formulas for Kx and Kz to multiply the vertical and horizontal hydraulic conductivity by two (Figure Box 7-2).

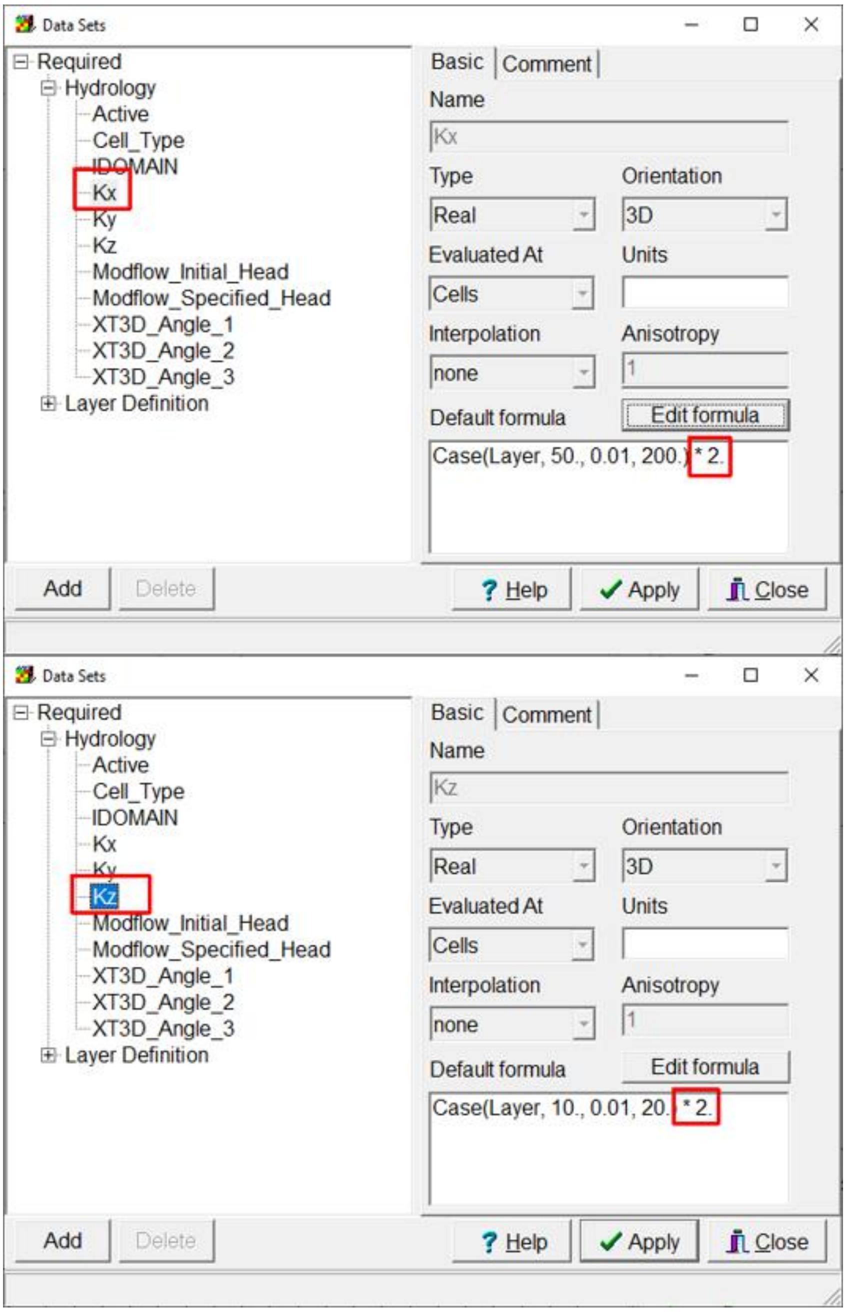


Figure Box 7-2 - Screen captures illustrating multiplying the hydraulic conductivities by two.

4. Save the ModelMuse file. Then export the MODFLOW input files and run MODFLOW. Import the model results when you are done.

Return to where text links to Box 7↥

Box 8 Step-by-Step Instructions for Model PS1B2

1. Open PS1A, modify the project description, and save as PS1B2.
2. Edit the two objects that define the specified head boundaries by slecting the red arrow button on the toolbar and then select and double click on the river CHD boundary. Raise the specified head to 321 m (Figure Box 8-1, top). Then select and double click on the canal CHD boundary and raise the specified head to 331 m (Figure Box 8-1, bottom).

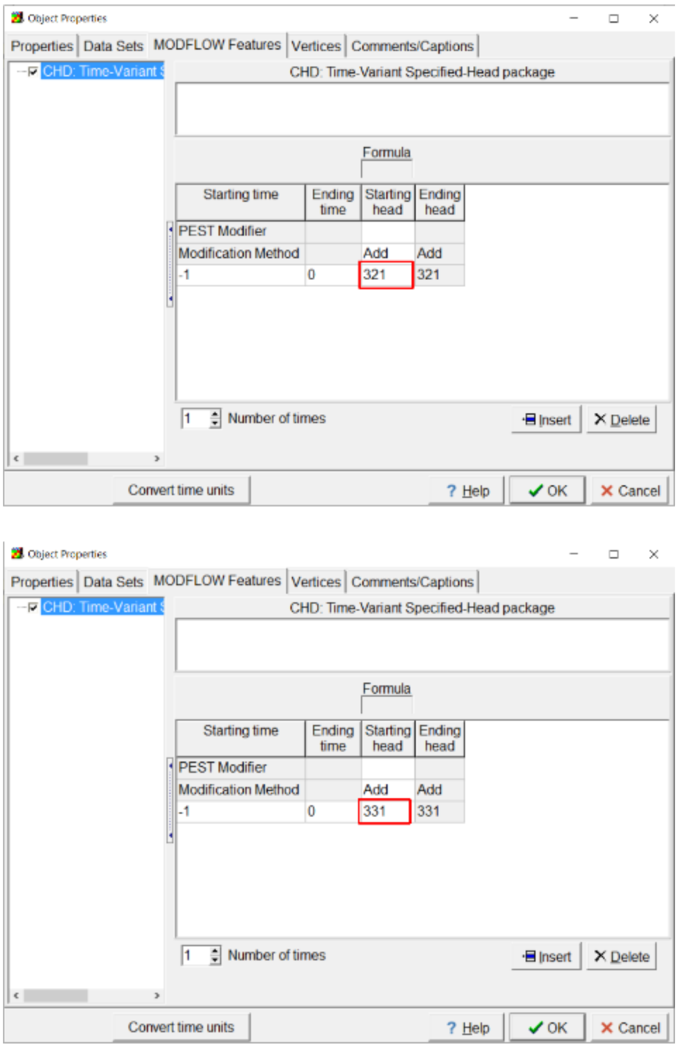


Figure Box 8-1 - Screen captures illustrating increasing the specified heads by one.

3. Save the ModelMuse file. Then export the MODFLOW input files and run MODFLOW. Import the model results when you are done.

Return to where text links to Box 8↑

Box 9 Step-by-Step Instructions for Model PS1B3

1. Open PS1A and save as PS1B3.
2. Modify the description in the *Model|MODFLOW Options* dialog box.
3. Modify the formulas for Kx and Kv to change the horizontal and vertical hydraulic conductivities of the silt layer (Figure Box 9-1).

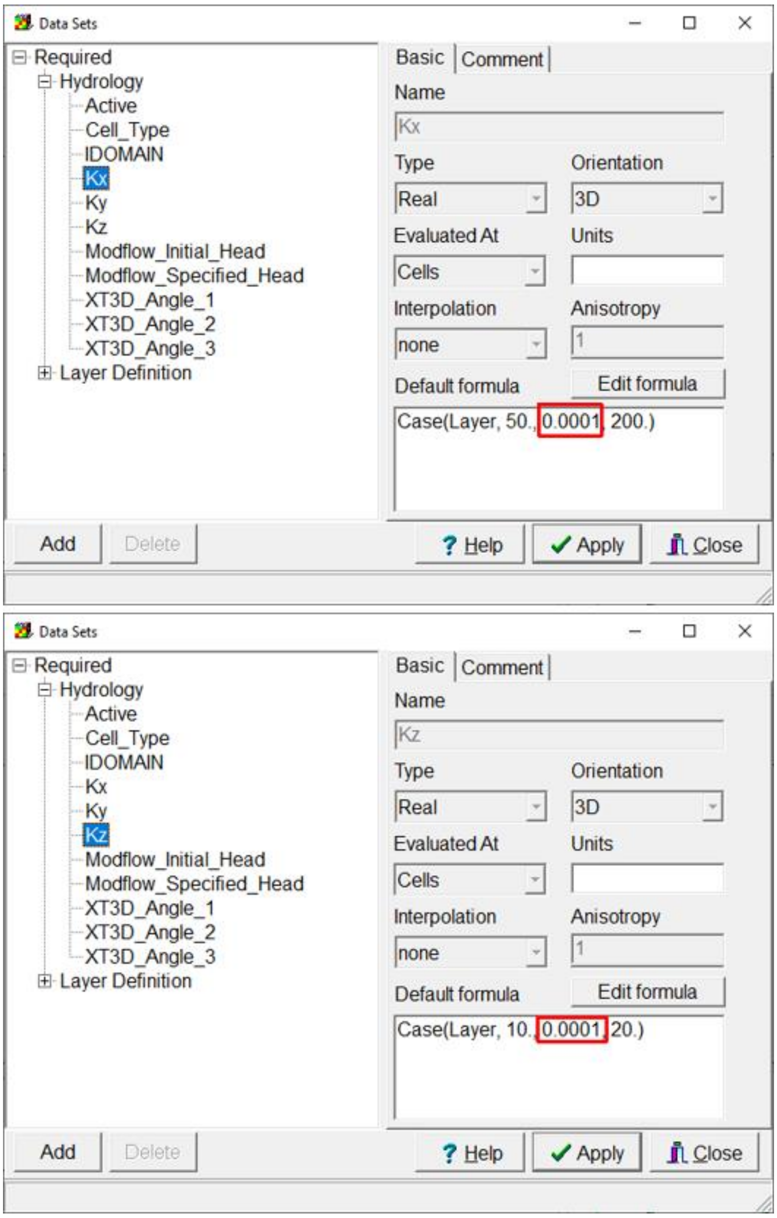


Figure Box 9-1 - Screen captures illustrating modifying the hydraulic conductivity of the silt layer.

4. Save the ModelMuse file. Then export the MODFLOW input files and run MODFLOW. Import the model results when you are done.

[Return to where text links to Box 9⮥]

Box 10 Step-by-Step Instructions for Model PS1B4

1. Open PS1A and save as PS1B4.
2. Create a new rectangular object that surrounds the cells in question by selecting *Object|Create|Rectangle* (Figure Box 10-1). On the Data Sets tab of the Object Properties dialog box, enter the value for Kx in the affected cells. Be sure the formulas for the higher and lower Z coordinates enclose just layer 1 (Figure Box 10-2).

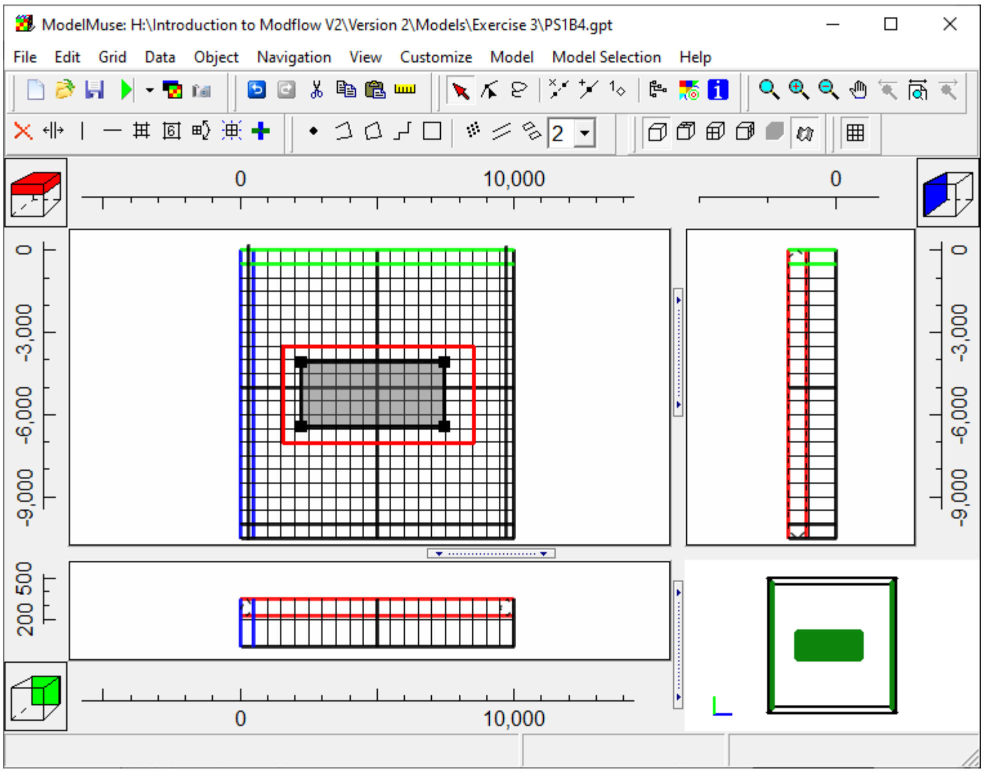


Figure Box 10-1 - Screen capture illustrating the position of an object that specifies a different value of hydraulic conductivity.

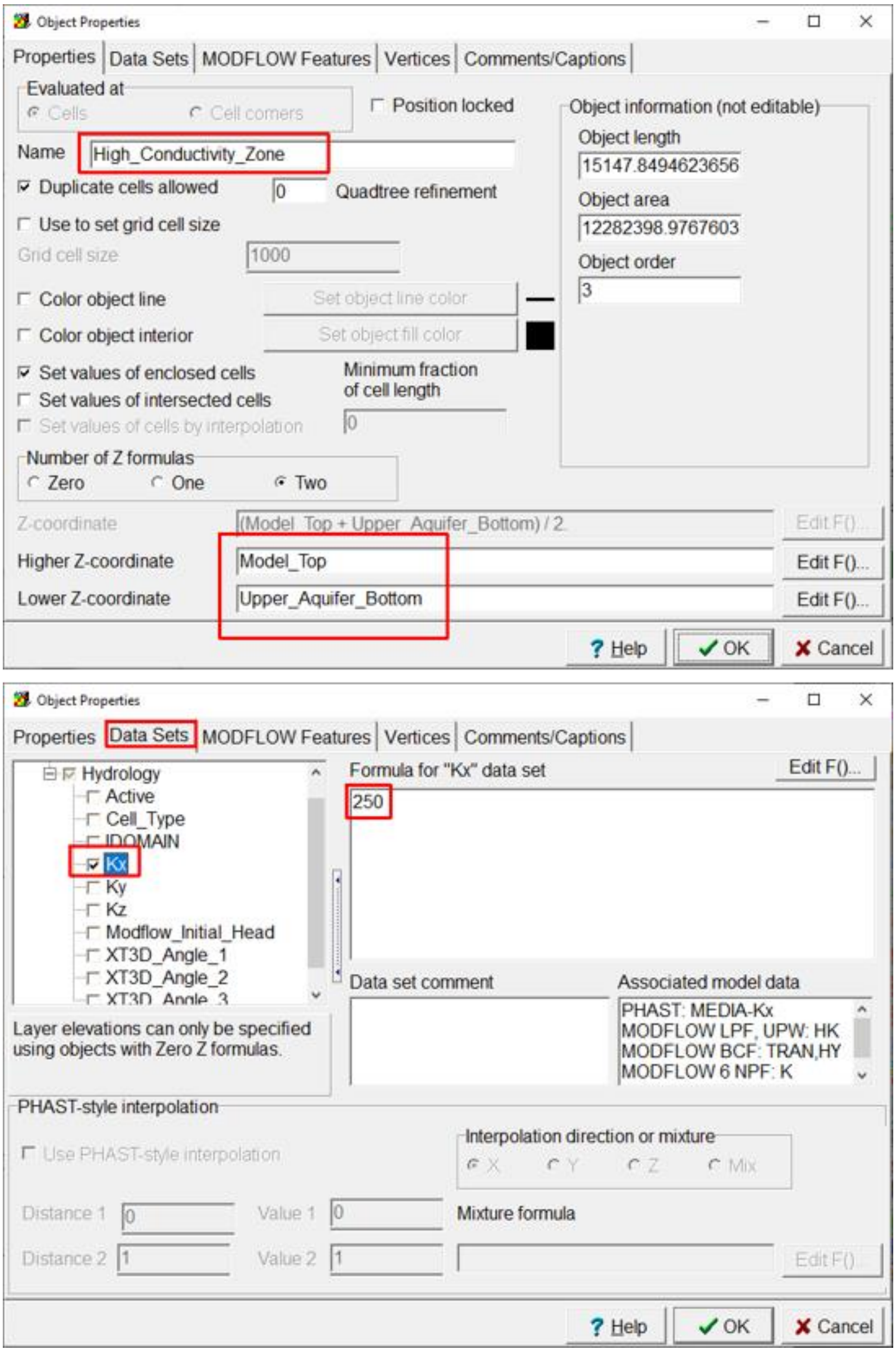


Figure Box 10-2 - Screen captures illustrating specifying the hydraulic conductivity in a limited zone.

3. Save the ModelMuse file. Then export the MODFLOW input files and run MODFLOW. Import the model results when you are done.

Return to where text links to Box 10↑

Box 11 Step-by-Step Instructions for PS2A

1. Open model PS1A (*File|Open*) and save it as PS2A (*File|Save As*). Change the description of the model in the *Model|MODFLOW Options* dialog box.
2. Select *Object|Select Objects* and click on the object that represents the canal on the western edge of the model to select it. Click the *Delete* button on the keyboard to delete the object.
3. Select *Model|MODFLOW Packages and Programs* and activate the Recharge package (under "*Boundary conditions|Specified flux*"). It is also a good idea to have the recharge apply to the top active cell so the recharge will enter the system even if a future simulation causes the top layer to go dry (Figure Box 11-1).

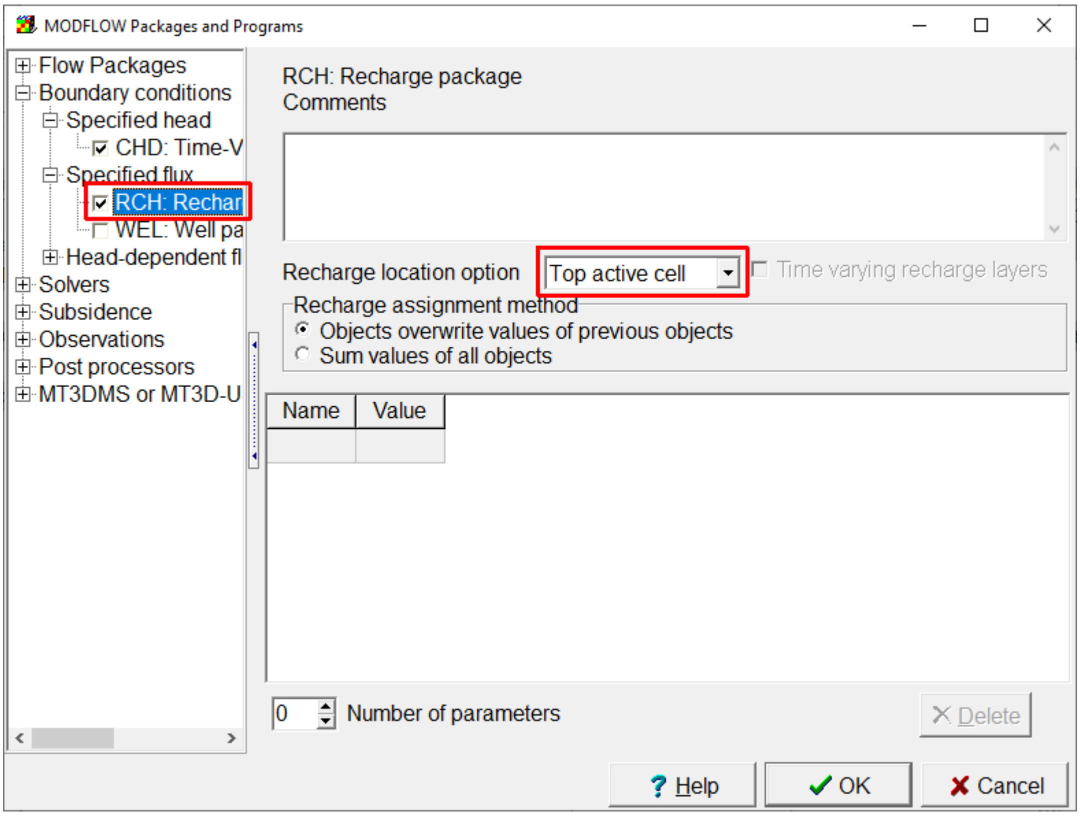


Figure Box 11-1 - Screen capture illustrating activation of the Recharge package.

4. A rectangular object to indicate recharge will be defined over the entire top surface of the grid. To do this, select *Object|Create|Rectangle* and click at one corner of the rectangle and then click again to define the opposite corner of the rectangle (Figure Box 11-2, top image). In the Object Properties dialog box, check the box next to RCH: Recharge package on the MODFLOW Features tab and enter the recharge rate (Figure Box 11-2).

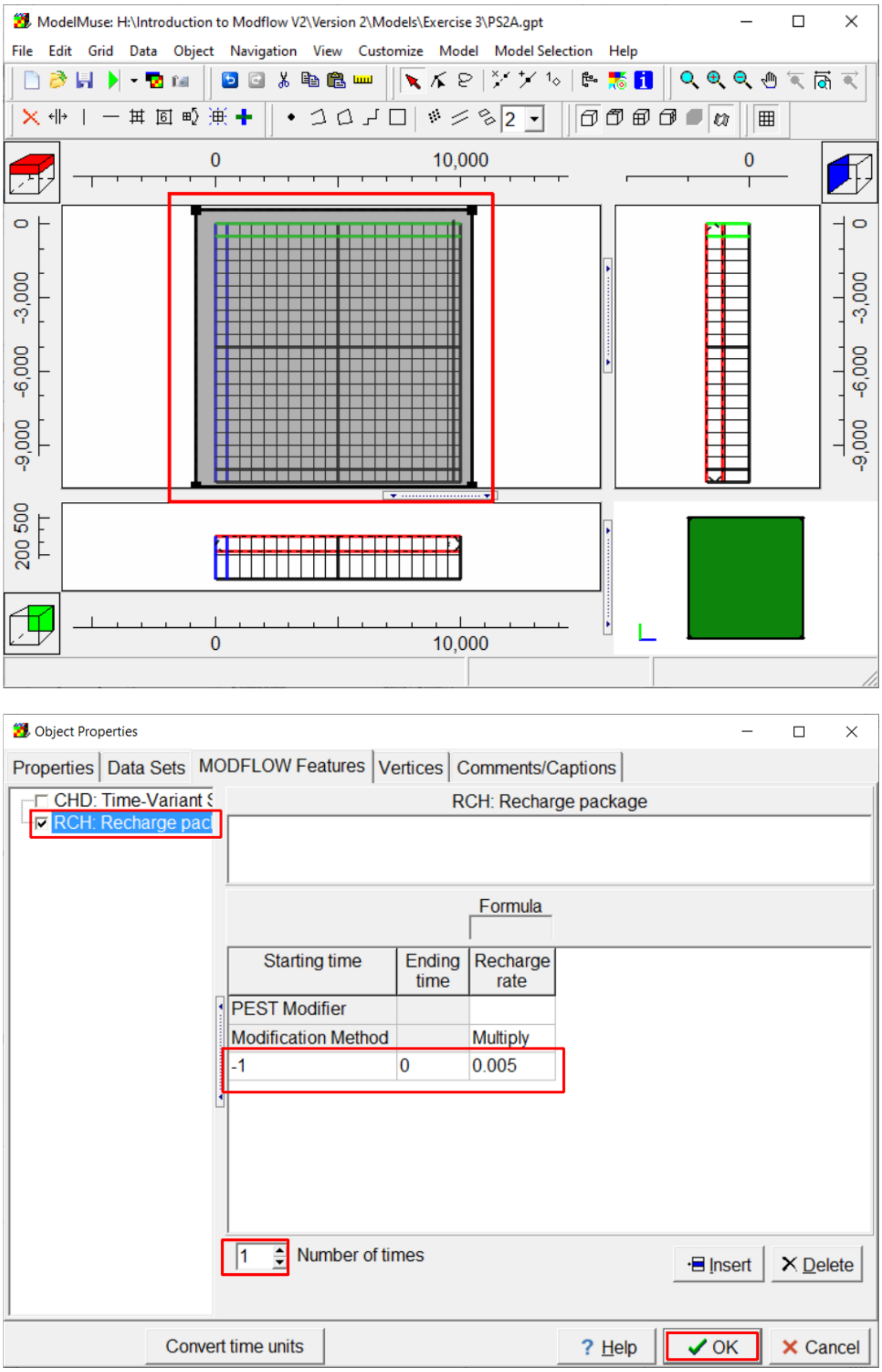


Figure Box 11-2 - Screen captures illustrating using an object to define the recharge boundary.

5. Select *File|Export|MODFLOW 6 Input Files* to run the model. To import the heads from the model, select *File|Import|Model Results*. For Display Choice, click *Neither*. After importing the model results, a cross section of the water table can be plotted by selecting *Data|Data Visualization* and selecting the Cross Sections pane. Then

expand down to *Data Sets|Optional|Model Results|Water Table|Water_Table_P1_S1* and click the red arrow pointing to the right to move that data set into the Data sets to plot table on the right. If desired, change the color assigned to the water table data sets by clicking on the color in the second column of the "Data sets to plot" table. Next, click *Apply* and *Close* (Figure Box 11-3). A thin line will be plotted across layer one from the west side of the model to the river in the panel at the bottom of the ModelMuse screen. The elevation of the water table line corresponds to the head elevations displayed in the main panel of the ModelMuse screen.

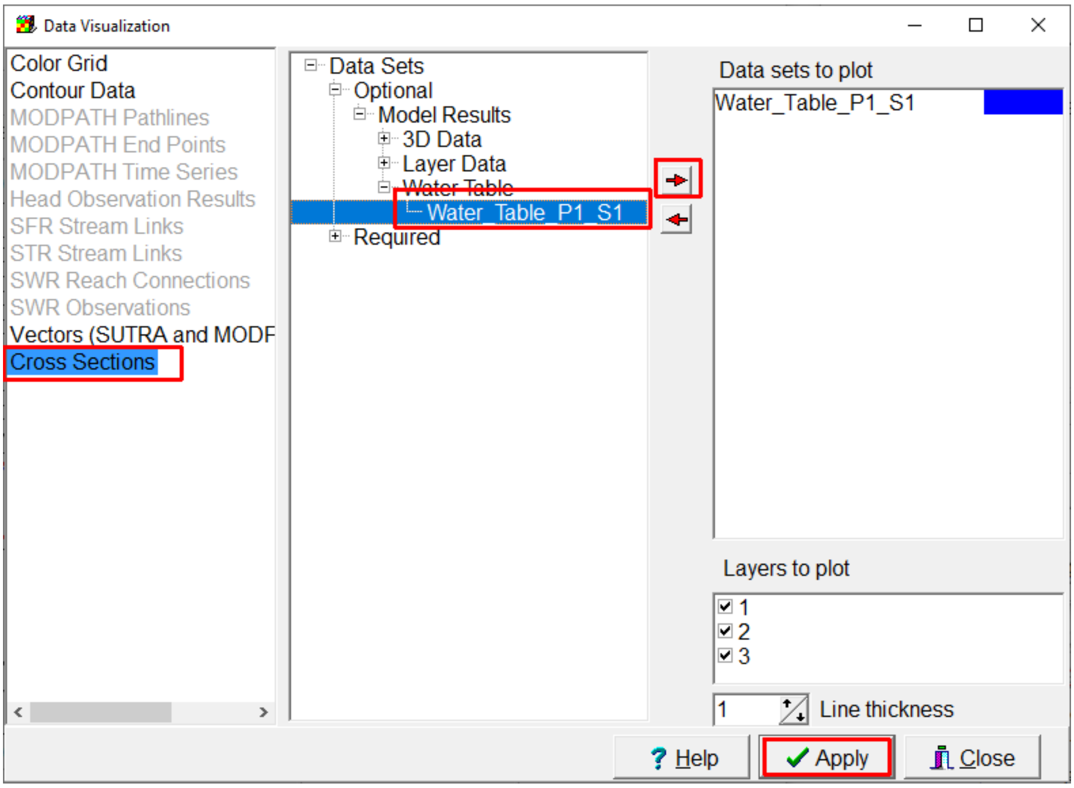


Figure Box 11-3 - Screen capture illustrating visualizing data on a cross section.

6. Save the model for use in the next exercise.

Return to where text links to Box 11

Box 12 Step-by-Step Instructions for PS2B

This model PS2B represents the river of model PS2A using the river package. The hydraulic conductivity of the riverbed is 20 m/d, and the width is 10 m. The elevation of the riverbed bottom is 317 m. The riverbed is 1 m thick.

1. Open PS2A and save it as PS2B. Change the description of the model.
2. Open the MODFLOW Packages and Programs dialog box and deactivate the CHD package and activate the River package as shown in Figure Box 12-1 (the River package is a head-dependent flux boundary).

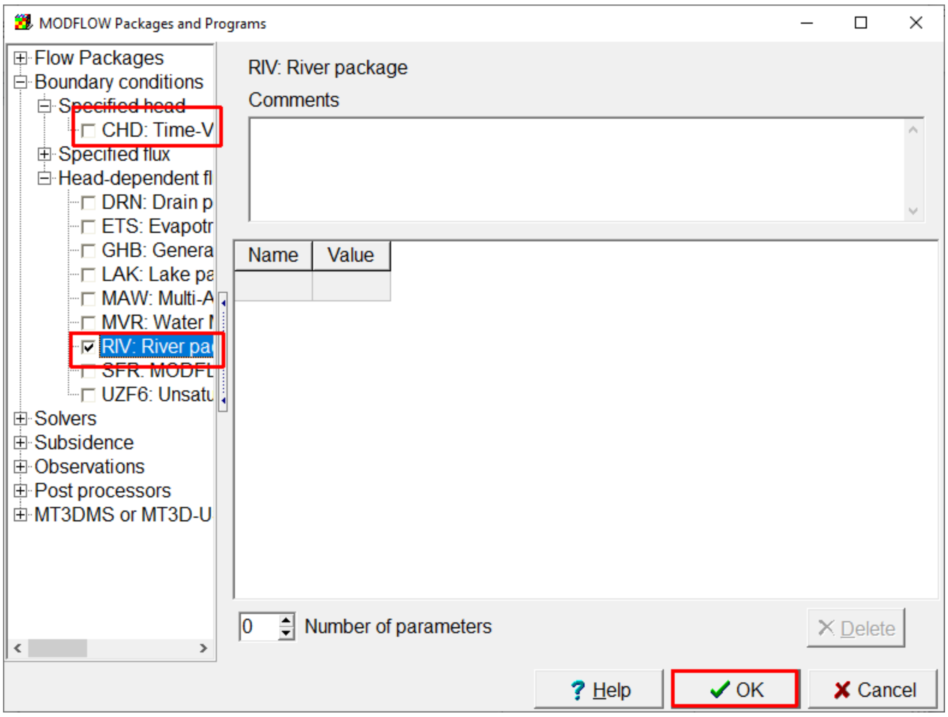


Figure Box 12-1 - Screen capture illustrating deactivating the CHD package and activating the River package.

3. Use global variables to define some of the properties of the river by selecting *Data|Edit Global Variables…* and defining global variables for the river width, river bed hydraulic conductivity, and the river bed thickness (Figure Box 12-2).

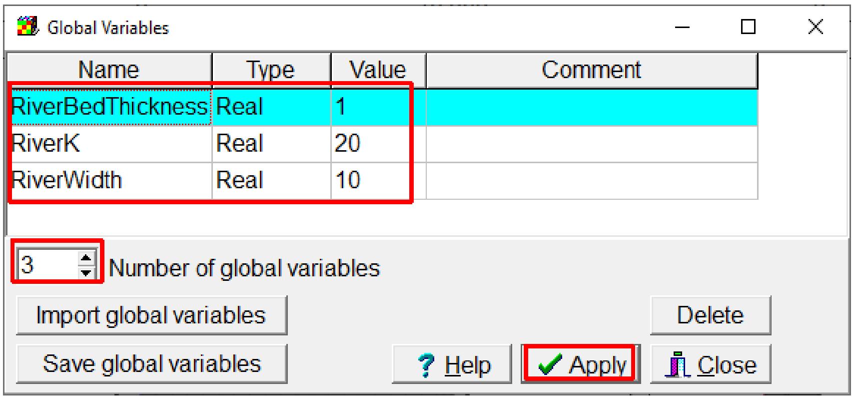


Figure Box 12-2 - Screen capture illustrating specification of global variables.

4. Select the object that defines the river on the eastern edge of the model and edit in the Object Properties dialog box. (Select *Object|Select Objects* and then double-click on the object. Another way to select it is to select *Object|Show or Hide Objects* and then double click on the appropriate object (Figure Box 12-3).

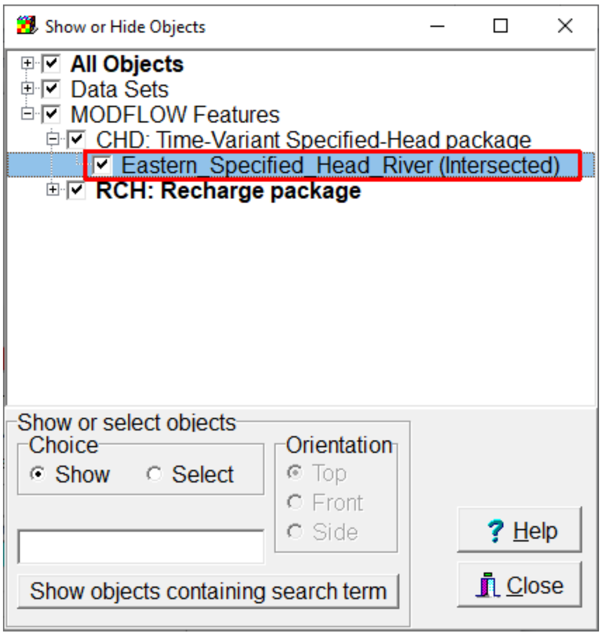


Figure Box 12-3 - Screen capture illustrating the use of the "Show or Hide Objects" dialog box to select objects for editing.

5. Check the check box for the River package on the MODFLOW Features tab.

6. Enter values for the starting and ending times, the river stage, and the river bottom (Figure Box 12-4). The conductance of the riverbed is equal to the area of the riverbed in the cell times the hydraulic conductivity of the riverbed divided by the thickness of the riverbed. The length of the river in the cell can be automatically calculated by ModelMuse. The length of the river in the cell times the river width is the area of the river in the cell. If we leave the conductance interpretation method set to "Calculated," whatever value we assign to the conductance per unit length will be multiplied by the length of the object within the cell. Thus, we can define the

conductance using a formula that includes the global variables for river properties. If you used the same names for the global variables as illustrated above, the formula would be "RiverK * RiverWidth / RiverBedThickness" (Figure Box 12-4). For this model, you can check the coordinates of the vertices of the object on the Vertices tab and make sure that the line goes exactly in North/South direction and goes completely through all the cells it intersects.

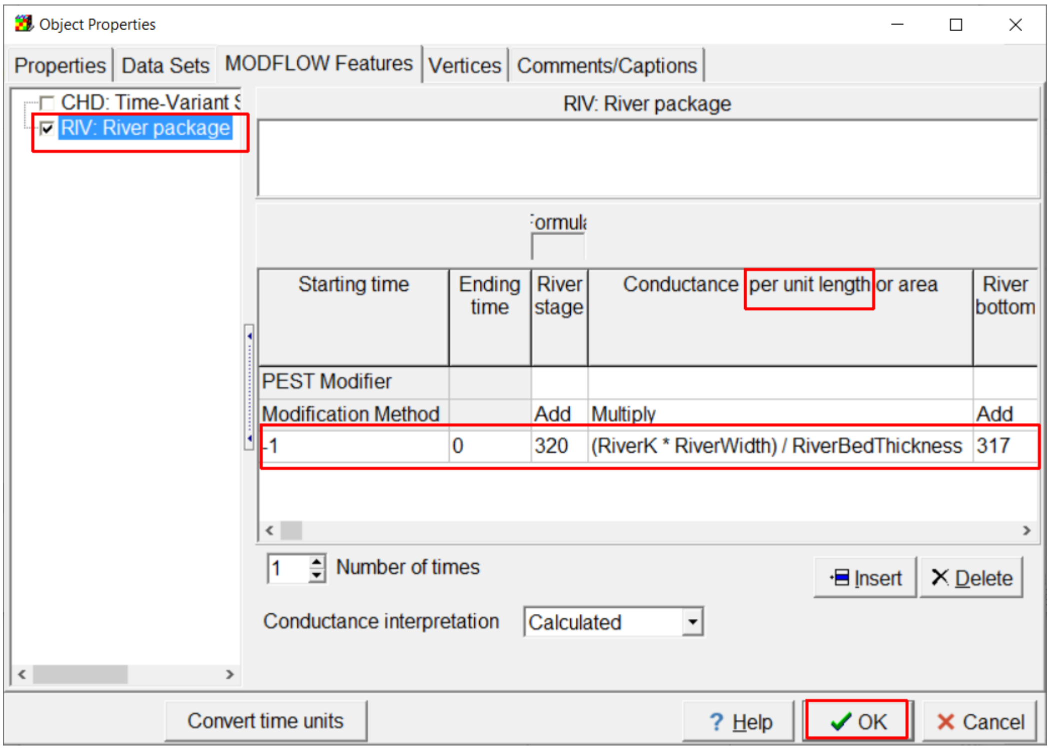


Figure Box 12-4 - Screen capture illustrating the use of a formula to specify the riverbed conductance.

7. Save and run the model, import the resulting heads, view the listing file, and answer the question for Exercise 5.10. Be sure to save the ModelMuse file for use in the next exercise.

Return to where text links to Box 12↑

Box 13 Step-by-Step Instructions for PS2C

This model is the same as model PS2B except that a buried drain tile, simulated with the DRAIN package, is installed in row 15 in columns 10-20. The conductance between the aquifer and the drain is 100,000 m^2/d, the elevation of the drain is 322.5 m.

1. Open the model PS2B and save it as PS2C. Edit the description of the model.
2. Open the MODFLOW Packages and Programs dialog box and activate the Drain package (Figure Box 13-1).

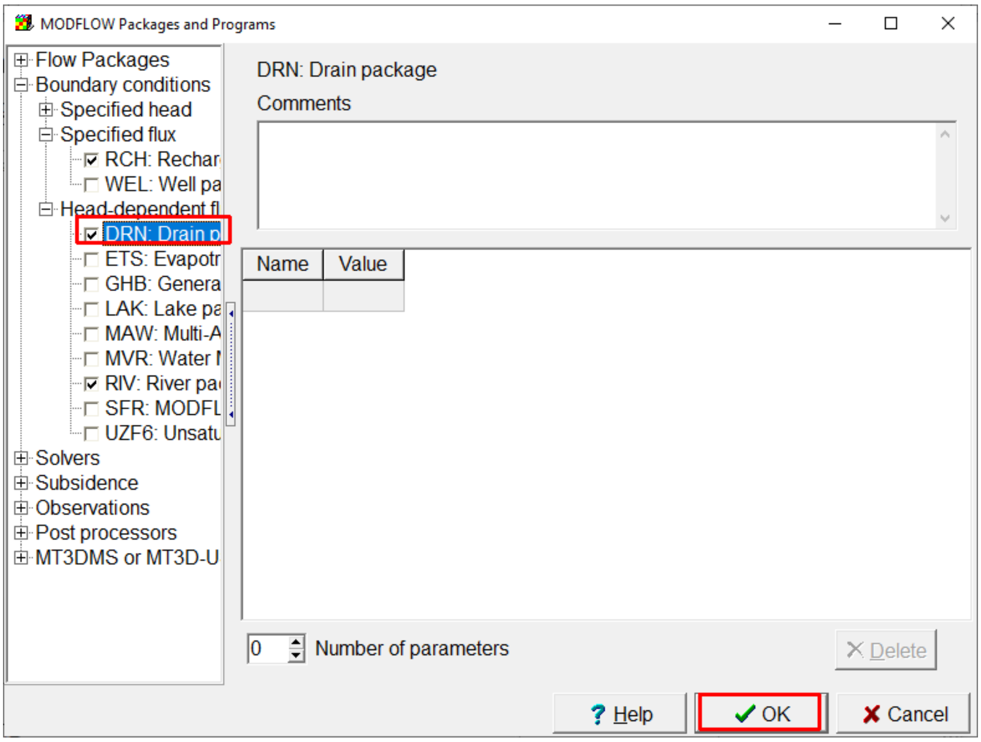


Figure Box 13-1 - Screen capture illustrating activating the Drain package.

3. Create a straight-line object along row 15 from columns 10 through 20 (Figure Box 13-2). In this case, the only thing that matters is that the object intersect the correct cells because the conductance will be specified directly rather than calculated.

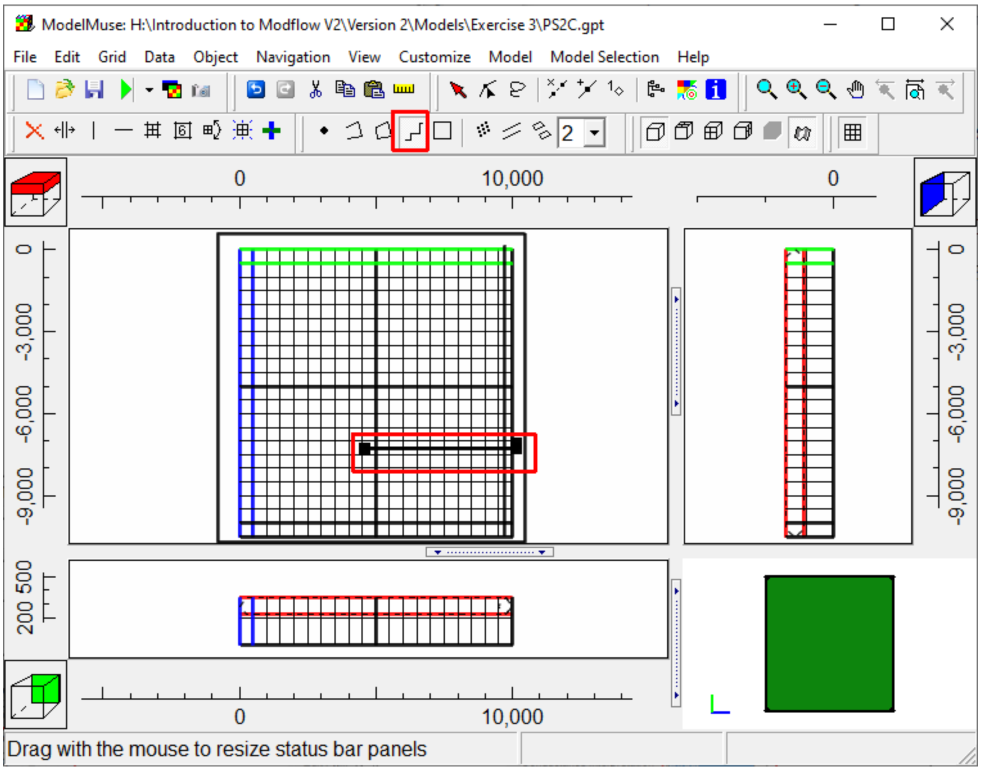


Figure Box 13-2 - Screen capture illustrating using an object to define drain boundaries.

4. In the Object Properties dialog box, make sure that the new object is in the top layer and activate the Drain package on the MODFLOW Features tab. In this case, we will specify the conductance directly rather than using a formula to specify the conductance. Thus, we set the conductance interpretation method to "Direct" and enter values for the starting and ending times, the drain elevation, and the drain conductance (Figure Box 13-3). If "Calculated" rather than "Direct" were selected, the modeler would specify the conductance per unit length and ModelMuse would calculate the conductance by multiplying the length of intersection between the object and the cell with the value specified by the modeler.

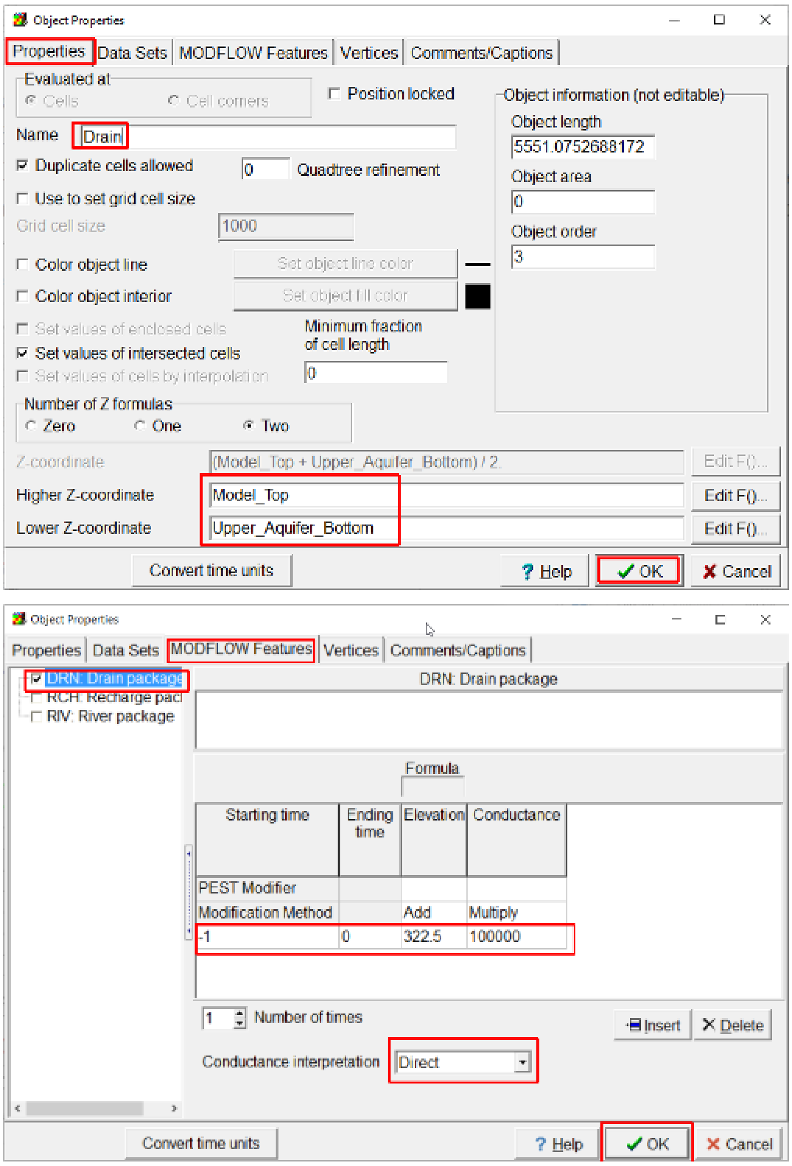


Figure Box 13-3 - Screen captures illustrating specifying the Drain properties.

5. Try coloring the grid with the drain conductance. Select *Data|Data visualization…*, make sure Color Grid is highlighted, and under "Data set or boundary condition" choose DRN Conductance. Notice that the conductance in all the drain cells is 100,000. If you wish, try changing the conductance interpretation to "Calculated" and notice the effect. You will probably see that not all the drain cells have the same conductance because the length of intersection of the object with the cells varies among the cells. After observing this effect, select *Edit|Undo* to revert to the previous value of the conductance interpretation.
6. Save the ModelMuse file. Then export the MODFLOW input files and run MODFLOW. Import the head results when you are done. Be sure to save the ModelMuse file for use in the next two exercises.

Return to where text links to Box 13↑

Box 14 Step-by-Step Instructions for PS2C1

1. Use *File|Save As* to save the PS2C model again as PS2C1. Change the description of the model.
2. In the MODFLOW Packages and Programs dialog box, activate the General Head Boundary (GHB) package and deactivate the Drain package (Figure Box 14-1).

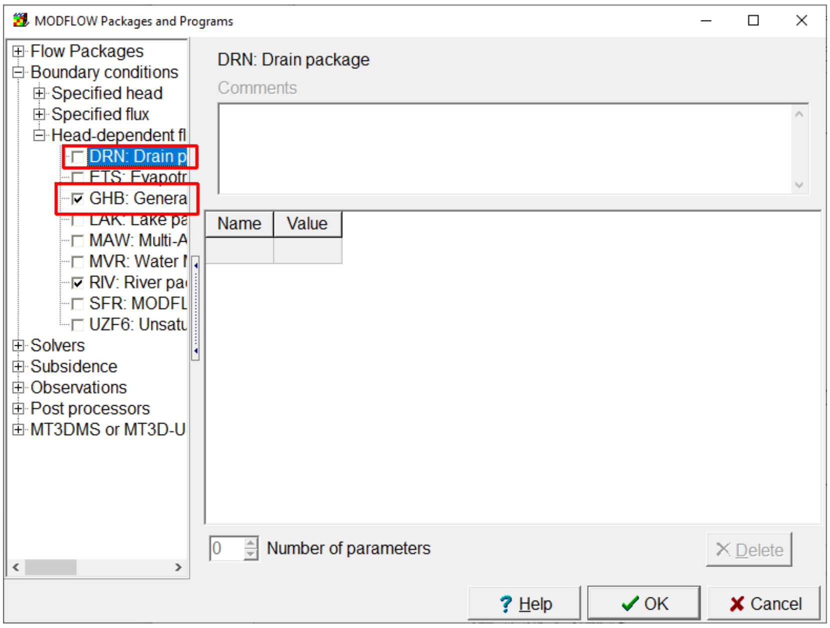


Figure Box 14-1 - Screen capture illustrating deactivating the Drain package and activating the General Head Boundary package.

3. Edit the object that formerly defined the drain and use it to define a General Head boundary (Figure Box 14-12).

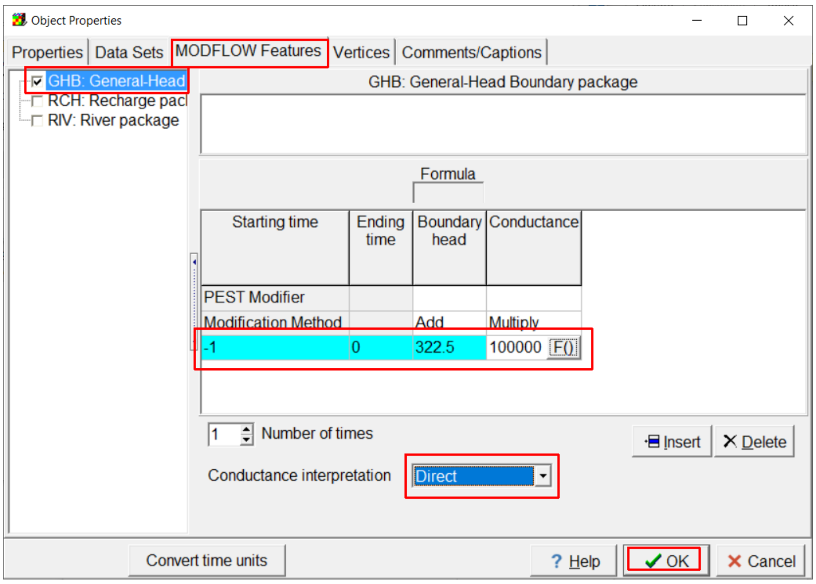


Figure Box 14-2 - Screen capture illustrating defining the properties of a General Head Boundary.

4. Save and run the model, import the resulting heads, view the listing file, and answer the question for Exercise 5.12. Compare the water budget for run PS2C1 (that uses the GHB) with the water budget for run PS2C (that uses DRN) and explain any difference you observe.

Return to where text links to Box 14↥

Box 15 Step-by-Step Instructions for PS2D

1. Open model PS2C and click *Save As* to create a new model named PS2D. Edit the description of the model.
2. In the MODFLOW Packages and Programs dialog box, activate the Well package (Figure Box 15-1).

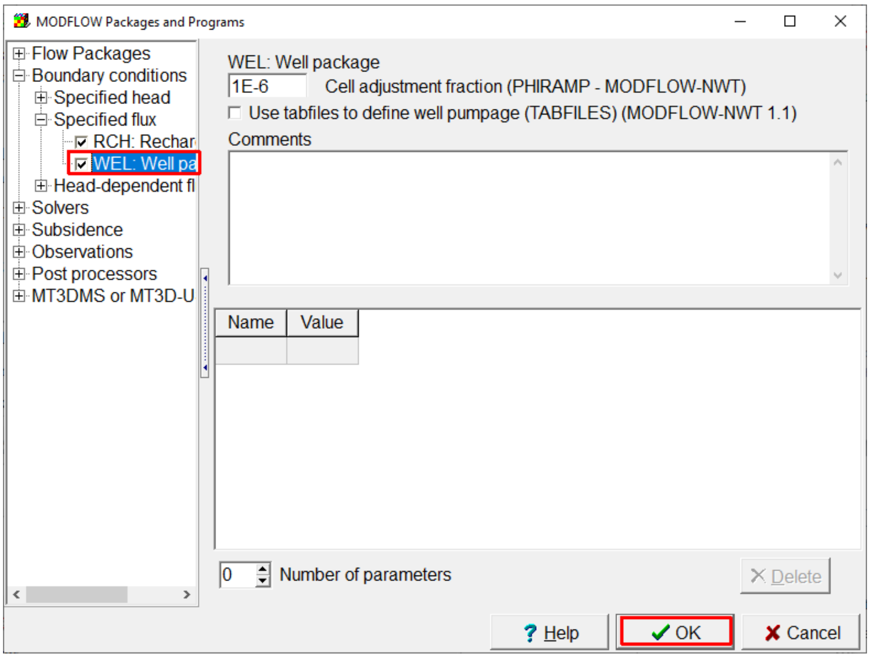


Figure Box 15-1 - Screen capture illustrating activation of the Well package.

3. Create a point object at row 11, column 10 (Figure Box 15-2).

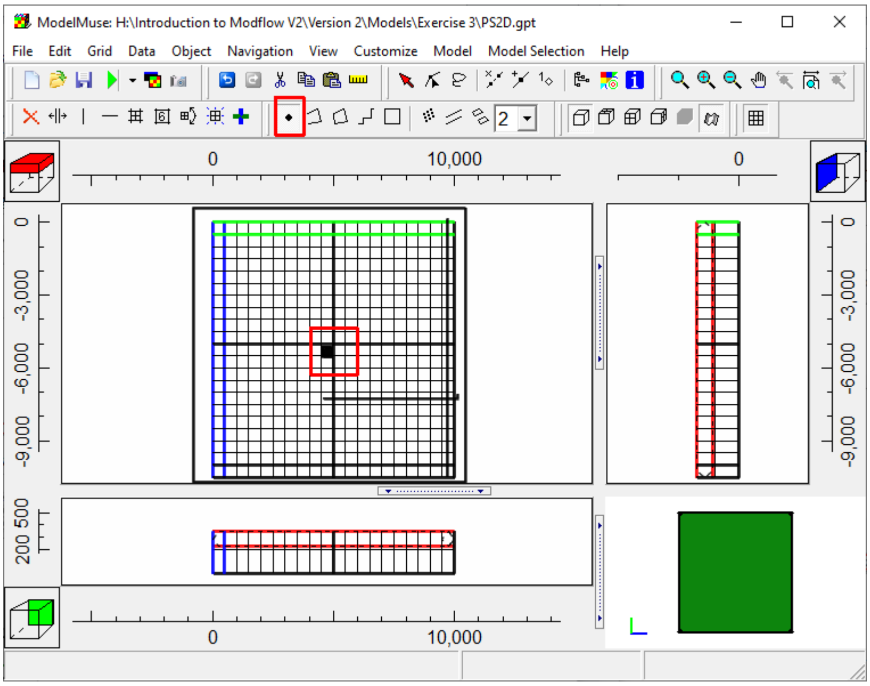


Figure Box 15-2 - Screen capture illustrating the use of a point object to define a well.

4. In the Object Properties dialog box, make sure that the object is in layer 1 and use it to define a well (Figure Box 15-3).

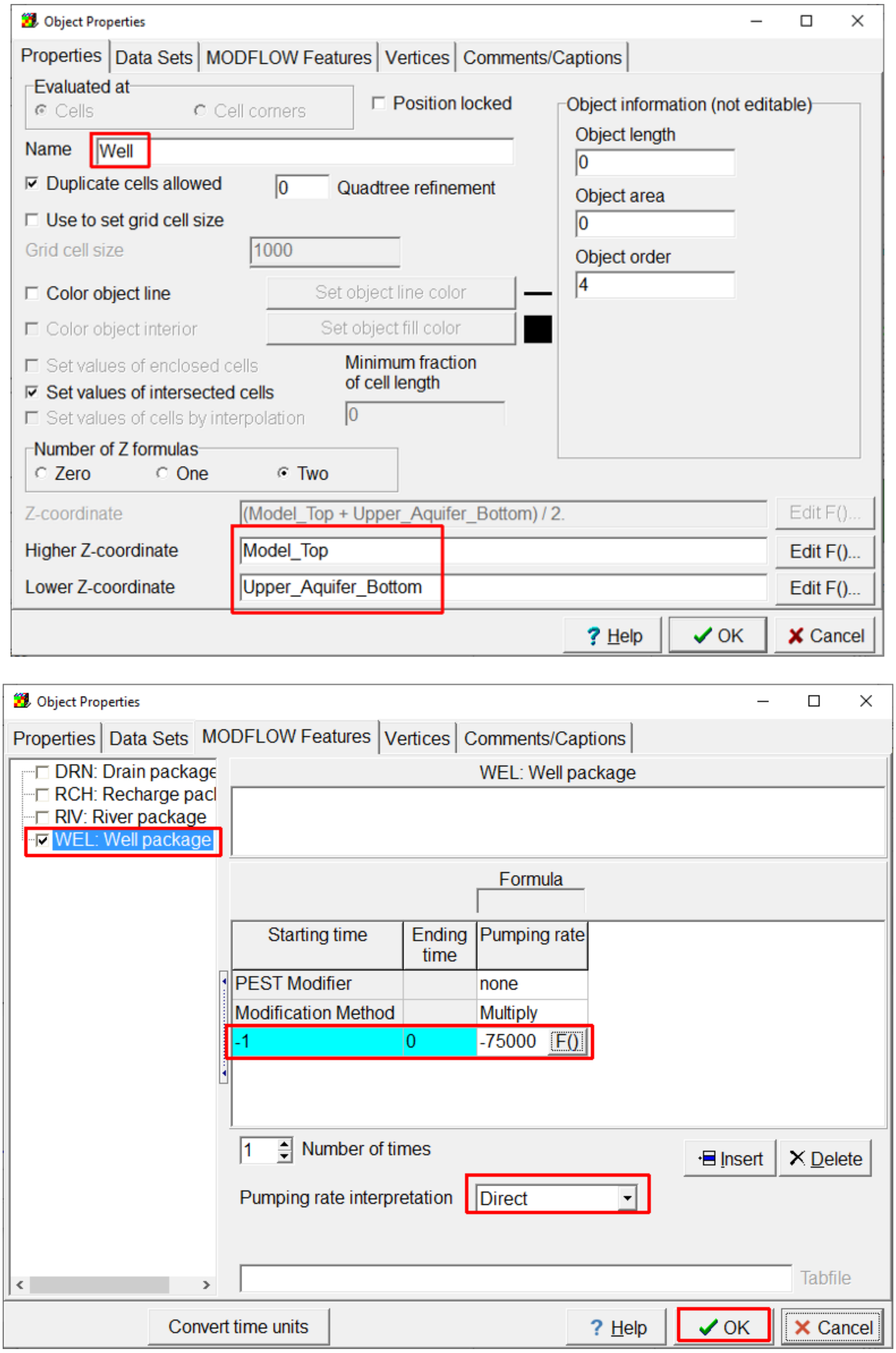


Figure Box 15-3 - Screen captures illustrating defining a well in the Object Properties dialog box.

5. Save and run the model.

6. Import the heads from the .bhd file (*File|Import|Model Results*) and see what effect the well has had on the water levels.
7. Be sure to save the ModelMuse file for use in the next exercise.

Return to where text links to Box 15↥

Box 16 Step-by-Step Instructions for PS2D1

1. Save the PS2D model again as PS2D1. Change the description of the model.
2. Change the selected layer to layer 3. One way to change the selected layer is to click on the Model Cube in the top left corner of the top view of the model. Another way is to select *Grid|Set Selected Col, Row, and Layer…* and enter the new selected layer number in the dialog box. Next, create a point object at coordinates (2374, -6237). (Figure Box 16-1). You can specify the exact coordinates on the Vertices tab of the Object Properties dialog box.

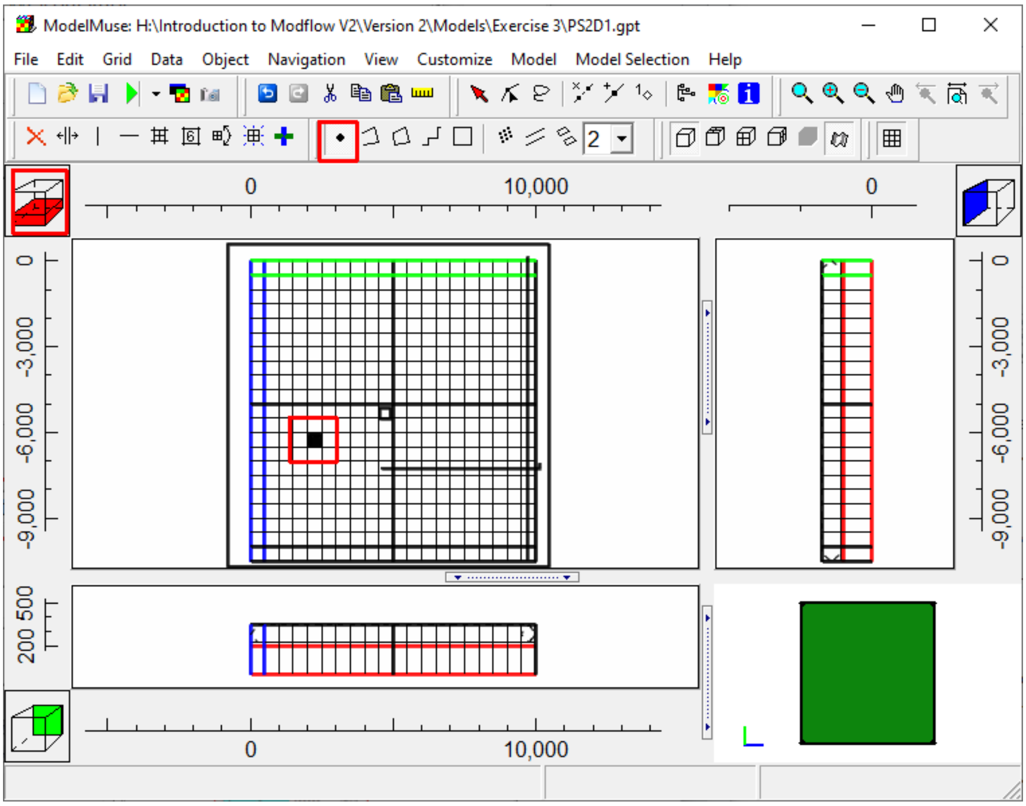


Figure Box 16-1 - Screen capture illustrating using another point object to define another well.

3. In the Object Properties dialog box, make sure the Z coordinates place the well in layer 3. On the MODFLOW Features tab, define a well with the required pumping rate (Figure Box 16-2).

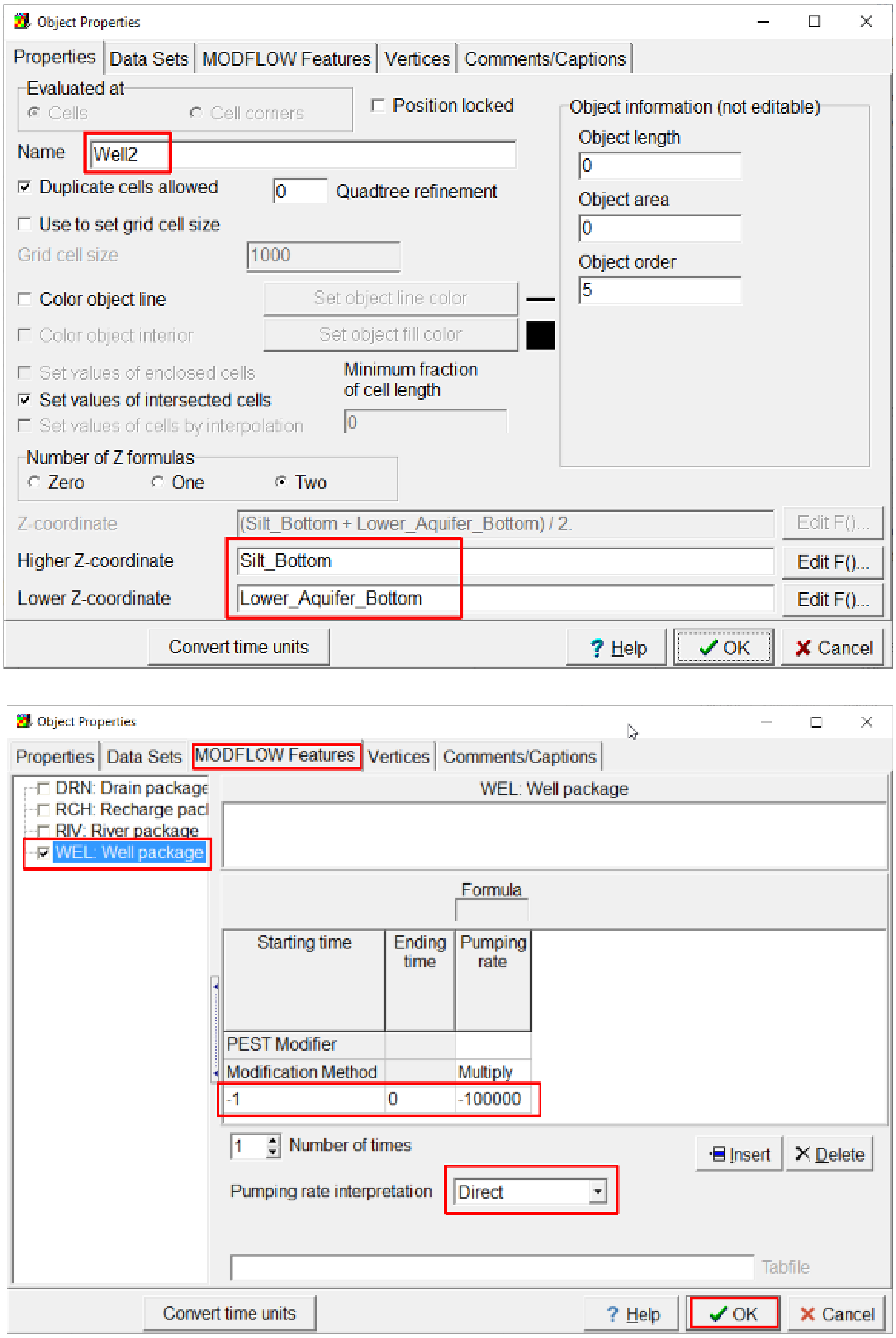


Figure Box 16-2 - Screen capture illustrating specifying the well properties in the Object Properties dialog box.

4. Select *File|Save* to save the model and then select *File|Export MODFLOW 6 Input Files* to run the model. When the model has finished running, select *File|Import Model Results* and select the binary head output file. Choose the contouring option (Figure Box 16-3).

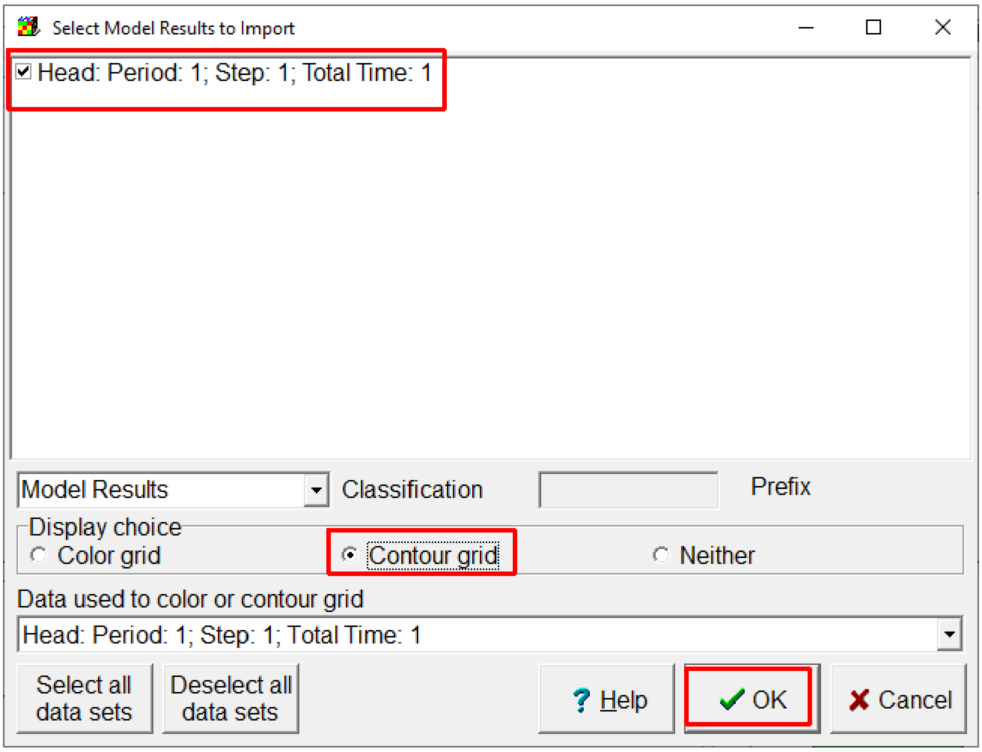


Figure Box 16-3 - Screen capture illustrating importing model results and contouring them.

5. You can change the selected layer to change the layer on which the contours are drawn.

Return to where text links to Box 16

Box 17 Step-by-Step Instructions for PS3Base

1. The reader can build this model based on information gleaned from previous exercises and name it PS3BASE, or start with the PS3Base.gpt provided in the folder of exercises on this book's web page↗. Load the model into a Graphical User Interface (GUI), examine its contents, and then run MODFLOW to generate output, import the calculated heads to visualize the head distribution, and view the results in the listing file.
2. Start ModelMuse and open the existing file PS3Base.gpt (Figure Box 17-1).

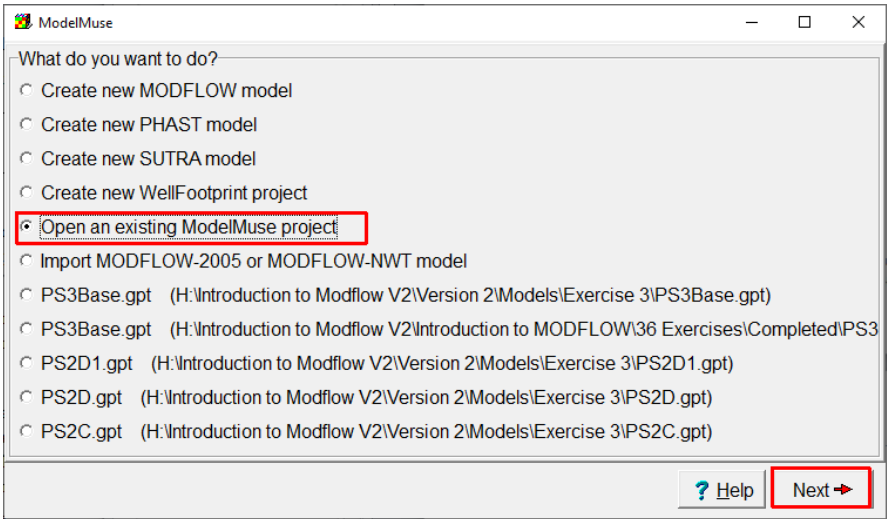


Figure Box 17-1 - Screen capture illustrating opening an existing model.

3. Run PS3Base (Figure Box 17-2).

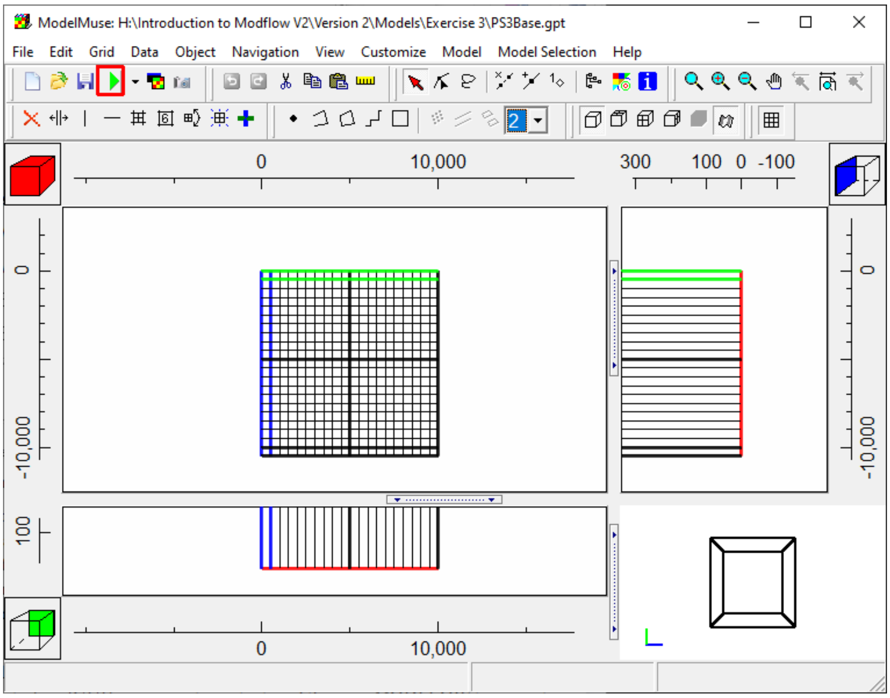


Figure Box 17-2 - Screen capture illustrating running PS3Base.

4. Import the simulated heads into your model by selecting *File|Import|Model Results* and select *PS3Base.bhd*, then in the dialog box, set the classification to User Defined and specify a prefix. By specifying the prefix, the data sets that will be imported will be easy to recognize later (Figure Box 17-3).

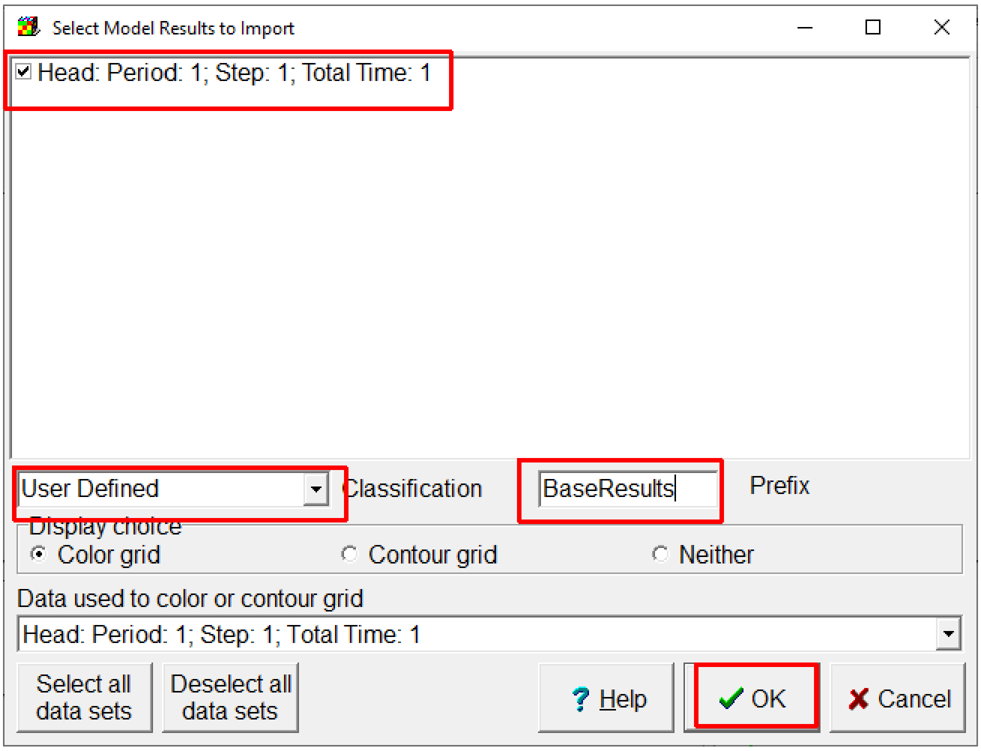


Figure Box 17-3 - Screen capture illustrating importing model results using a specific prefix.

5. Save the model for reuse in parts A and B.

Return to where text links to Box 17

Box 18 Step-by-Step Instructions for PS3A

1. Open PS3Base in ModelMuse. Select *File|Save As* and save it as PS3A.
2. Open model PS3BASE in a GUI and use the *Save As* option to create a new model named PS3A.
3. Select *Model|MODFLOW Time…* to provide input to the TDIS package. Define three stress periods with the following properties as shown in Figure Box 18-1.
    - Stress period 1: length = 1, time steps = 1, multiplier = 1, type = SS (steady state)
    - Stress period 2: length = 1800, maximum first time-step length = 16, multiplier = 1.5, type = TR (transient)
    - Stress period 3: length = 1, time steps = 1, multiplier = 1, type = SS (steady state)

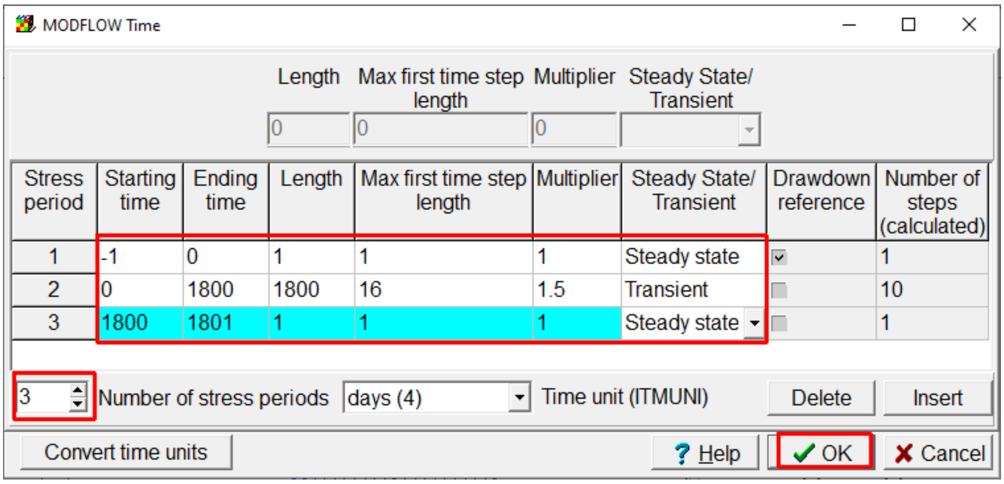


Figure Box 18-1 - Screen capture illustrating specifying model stress periods.

4. When you close the dialog box, you will get a message that the storage package has been activated. Activating the Storage package caused data sets for specific yield and specific storage to be created. Be sure that specific yield and specific storage have been specified in the *Data|Edit Data Sets* dialog box.
    - Specific yield = 0.2
    - Specific storage = 0.000001 m^{-1}
5. Make the changes required to specify constant heads along the north and south boundaries (row 1 and row 21) as delineated in steps 6 through 8.
6. Select *Model|Packages and Programs* and activate the CHD package Figure Box 18-2).

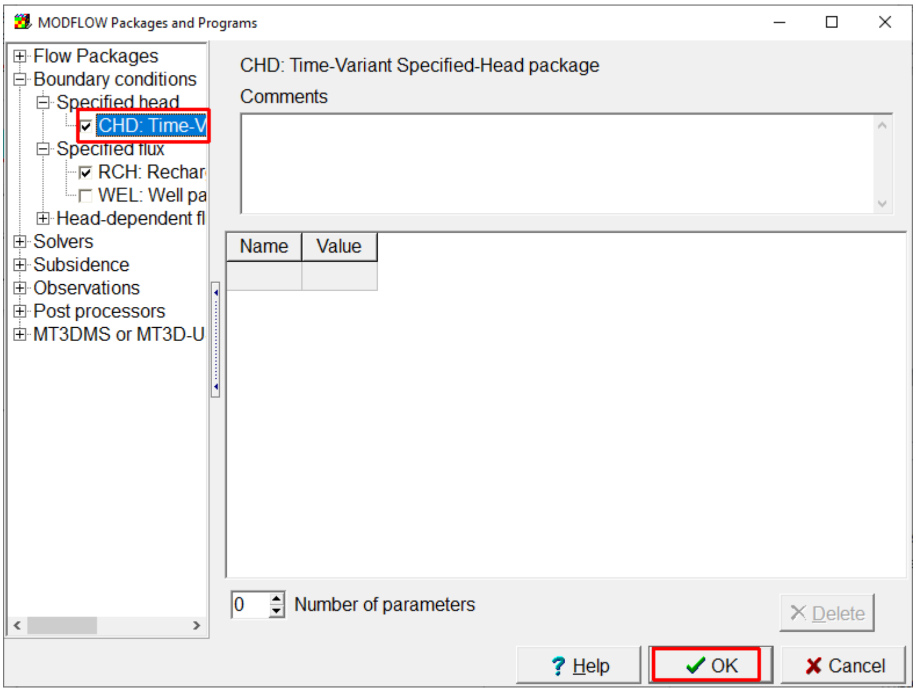


Figure Box 18-2 - Screen capture illustrating activating the Constant Head package.

7. Select *Object|Create|Straight Line* and draw a straight-line object in the top view of the model in the first row. Activate the CHD package in the Object Properties dialog box. Specify the starting and ending times to encompass all three stress periods and specify the head as the head imported from the base run (Figure Box 18-3). Repeat for the last row.

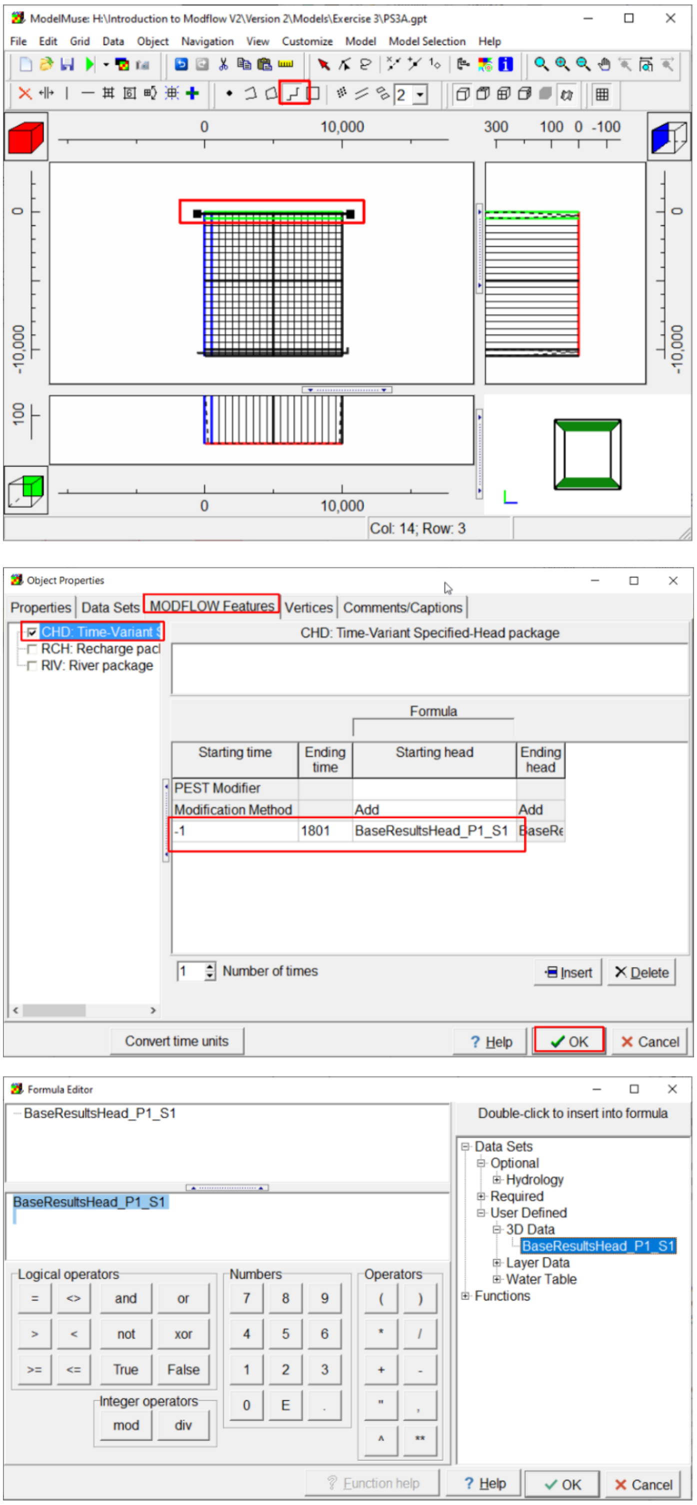


Figure Box 18-3 - Screen capture illustrating using an object to define specified head boundaries

8. Change starting head to be read from file PS3BASE.bhd. The format must be (BINARY). Select *Model|MODFLOW Options* and on the Options tab, select the head file from the base run (Figure Box 18-4). Making this change will result in an altered Initial Conditions package input file.

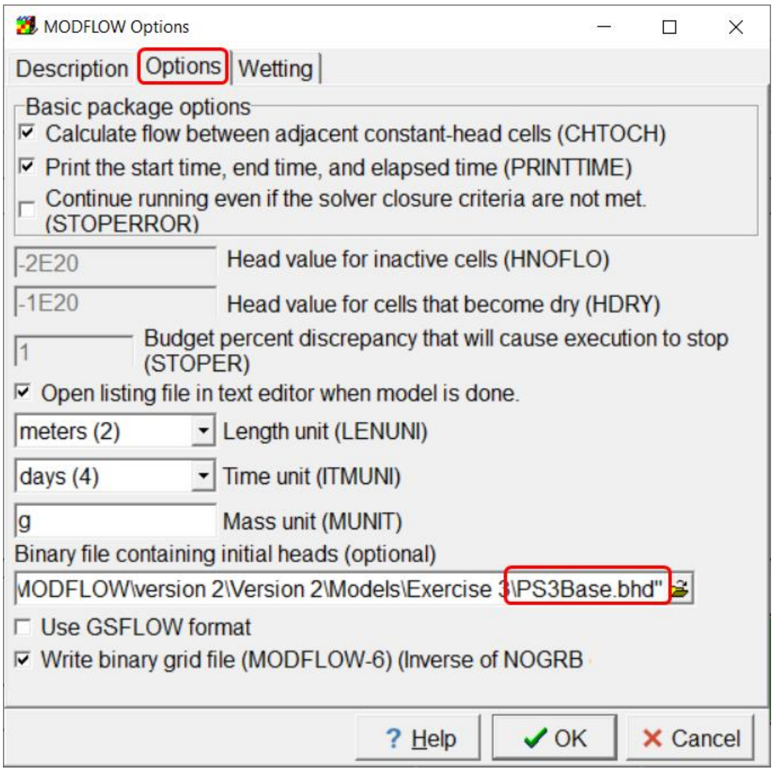


Figure Box 18-4 - Screen capture illustrating using the heads from the base model as the starting heads in the new model.

9. For stress periods 2 and 3, add a well to row 11, column 10 (coordinates = 4750±250, -5250±250) with a discharge rate of -100,000 m^3/d (the well is not present in stress period 1) as described in steps 10 and 11.
10. First activate the Well package in the *Model|MODFLOW Packages and Programs* dialog box (Figure Box 18-5).

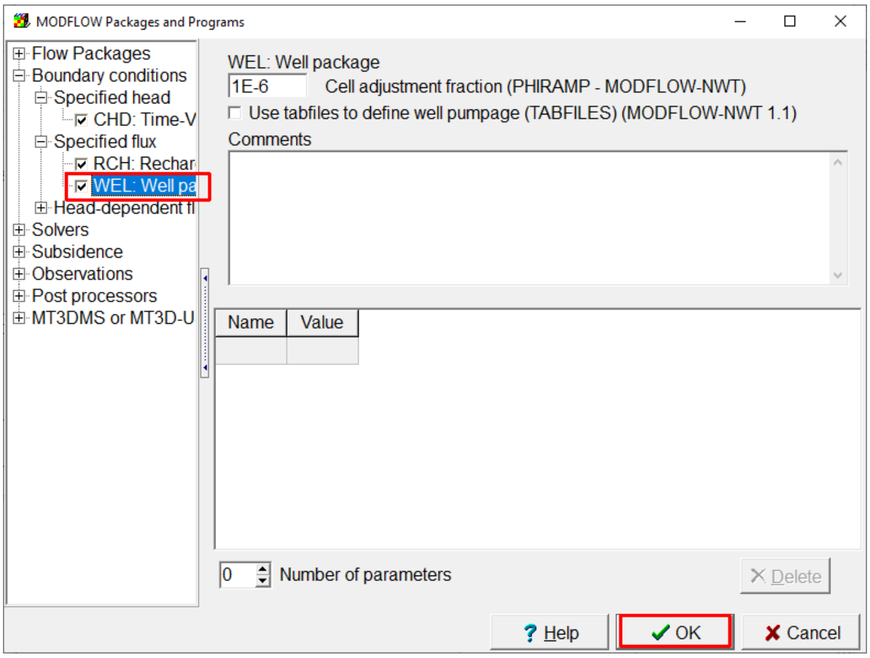


Figure Box 18-5 - Screen capture illustrating activating the Well package.

11. Next, create a point object—*Object | Create | Point*—and define the well properties in the Object Properties dialog box. Be sure that the times encompass only the second and third stress periods (Figure Box 18-6). The model has only one layer, so there is no need to specifiy the vertical position of the well.

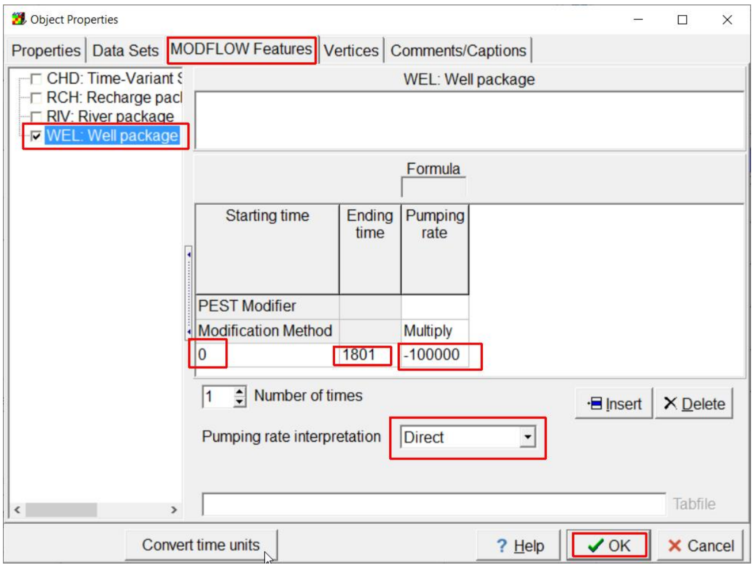


Figure Box 18-6 - Screen capture illustrating specifying the properties of a well that is active in the second and third stress periods.

12. Make the necessary modifications in the RIVER and RECHARGE packages to set all their properties to the same values for all three stress periods as described in step 13.
13. Open the River object and in the Object Properties dialog box and change the ending time (Figure Box 18-7). Repeat this for the recharge.

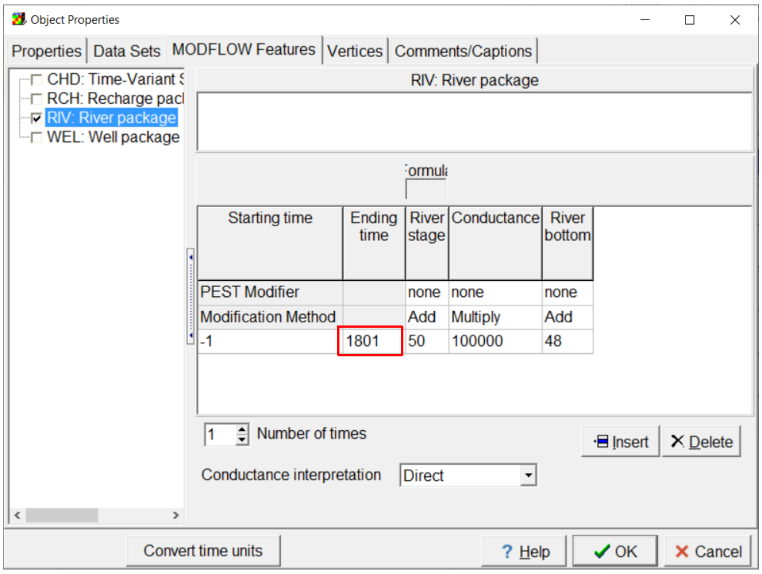


Figure Box 18-7 - Screen capture illustrating modifying the ending time of a boundary condition.

14. Modify the MODFLOW Output Control file to save and print head at all time steps as described in step 15. Any heads that are saved in an output file can be displayed in a graphical user interafce.
15. Select *Model|MODFLOW Output Control* and select *Head*. If you imported the model, the existing head options should already ensure that heads are saved at the end of every time step (Figure Box 18-8).

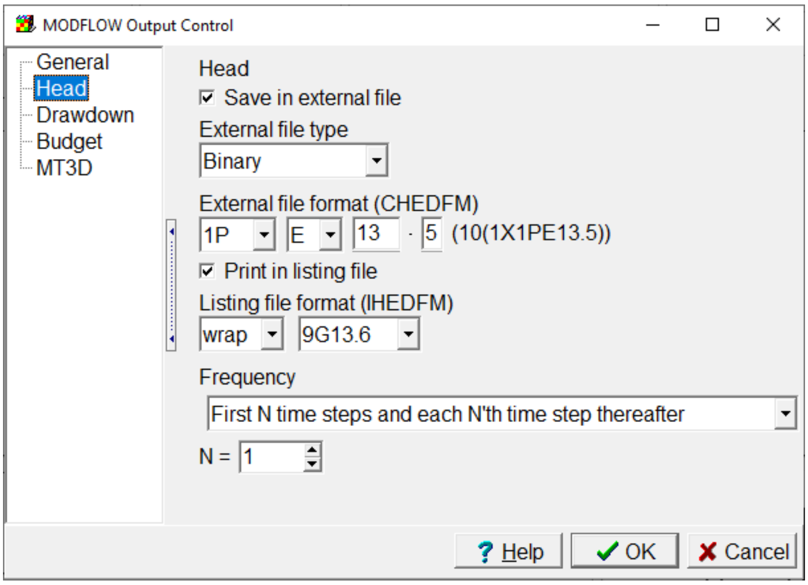


Figure Box 18-8 - Screen capture illustrating specifying the output control values.

16. Next export the MODFLOW input files and run MODFLOW. When ModelMuse creates the MODFLOW input files, it will detect that there are no active wells in the first stress period. Because it is unusual for a stress package to be active in a model but not be used in a stress period, ModelMuse issues a warning. You may ignore this warning in this case because the absence of wells in the first stress period is by design.
17. While ModelMonitor is running MODFLOW, it will look a little different than before. The graph of percent discrepancy will be plotted for each time step (Figure Box 18-9).

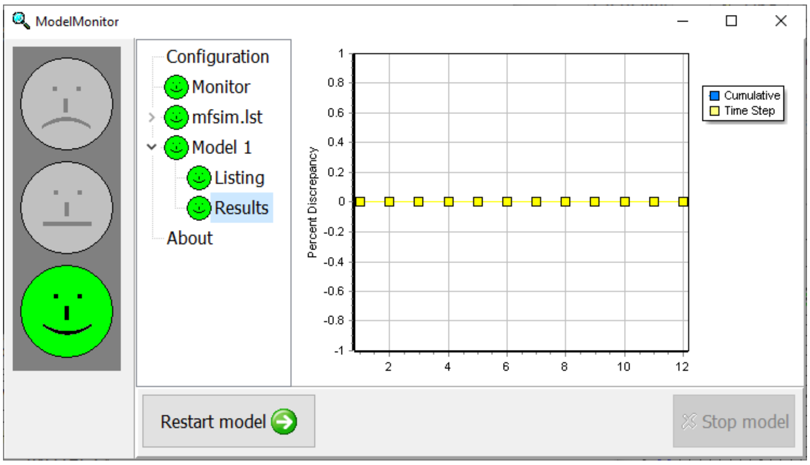


Figure Box 18-9 - Screen capture of the ModelMonitor window displaying the percent discrepancy in the MODFLOW water budget in each time step.

18. After MODFLOW has finished running, the heads can be imported for every time step for which they were saved. To import the heads for all the time steps, check the checkboxes for all of them or click the *Select all data sets* button (Figure Box 18-10).

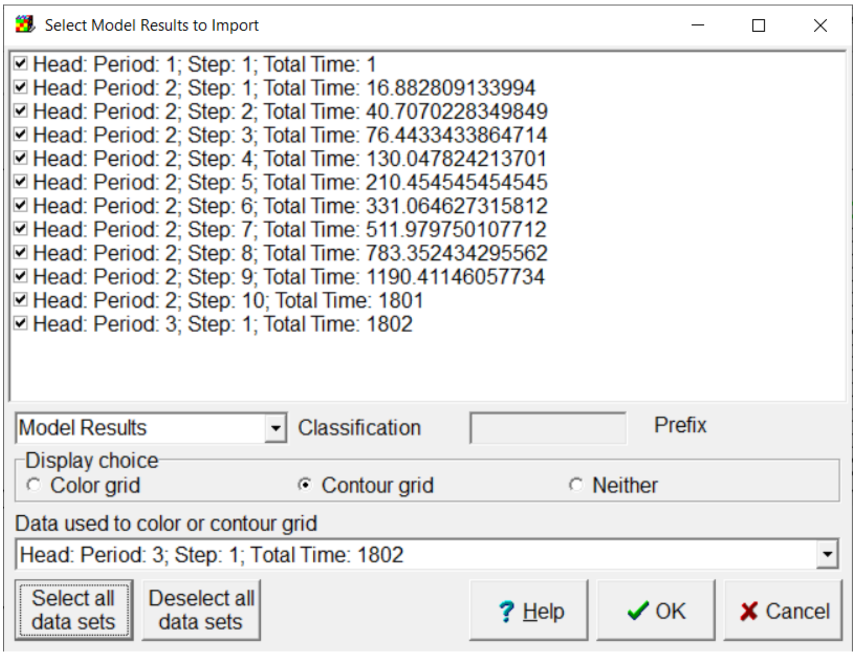


Figure Box 18-10 - Screen capture of the Select Model Results to Import dialog box with all the data sets for the model selected.

19. The model results for any particular time step can be viewed by selecting the desired data set in the *Data|Data Visualization* dialog box.
20. It is possible to view an animation of the model results in the *File|Export|Image* dialog box. To do so, open the Animation pane and check the check box for the 3D Data under Model Results and press the *Preview* button (Figure Box 18-11). You can close this box when done, without saving any images.

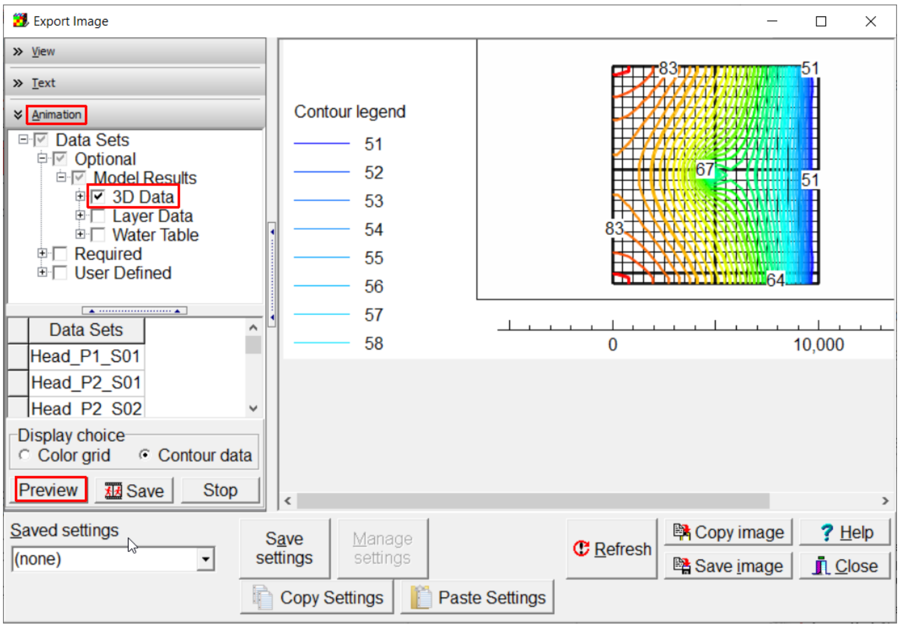


Figure Box 18-11 - Screen capture illustrating how to animate model results in ModelMuse.

Return to where text links to Box 18

Box 19 Step-by-Step Instructions for PS3B

1. Open PS3A.
2. Select *Model|MODFLOW Packages* and deactivate the CHD package (Figure Box 19-1).

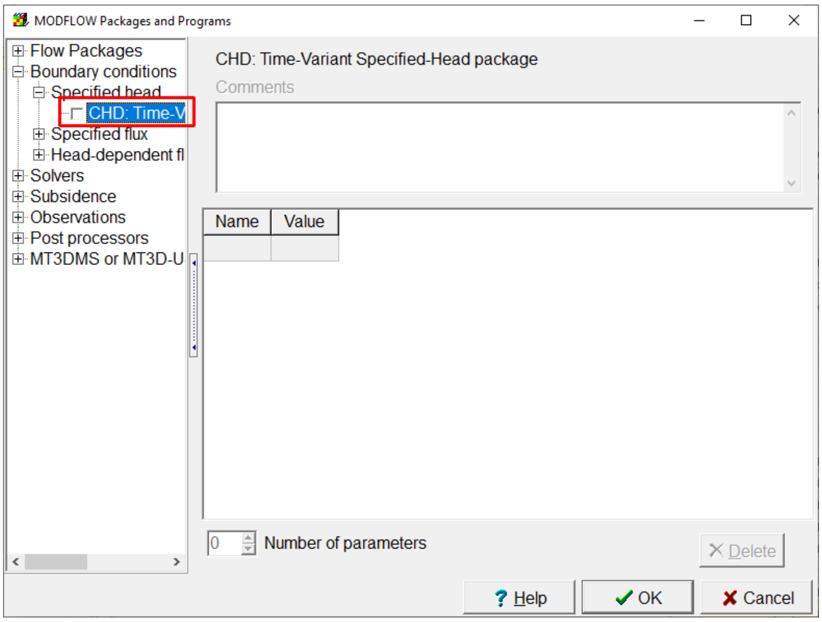


Figure Box 19-1 - Screen capture illustrating deactivating the CHD package.

3. Because the CHD package is deactivated, the objects that formerly defined the specified head boundaries no longer play that role although they retain data for the CHD package. They can be repurposed to define inactive cells. Open them in the Object Properties dialog box and use them to set the Active data set to False (Figure Box 19-2).

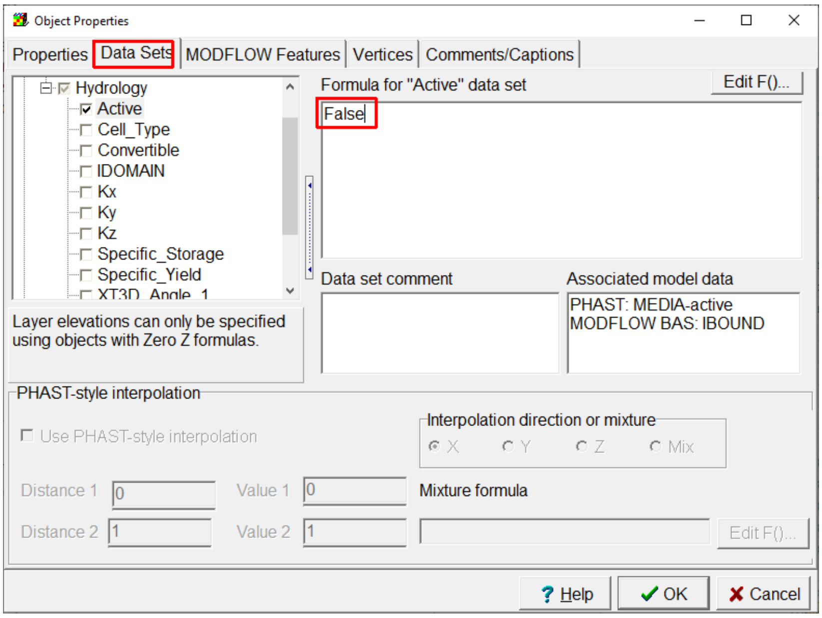


Figure Box 19-2 - Screen capture illustrating the use of object to deactivate cells.

4. Save the ModelMuse file. Then export the MODFLOW input files and run MODFLOW. Import the model results when you are done.

Return to where text links to Box 19↥

Box 20 Step-by-Step Instructions for PSMP

1. Open PSMP.gpt in ModelMuse.
2. Make sure that MODPATH version 7 is installed on your computer. Make sure that heads and flows will be saved for each time step by opening the *Model|MODFLOW Output Control* dialog box. On the Head pane, make sure the *Save in external file* checkbox is checked, the frequency is set to *First N time steps and each N'th time step thereafter* and N is set to 1. On the Budget pane, set the same frequency and N. Make sure that Save cell flows is set to Binary.
3. Select *Model|MODFLOW Packages and Programs* and activate MODPATH 7 (Figure Box 20-1).

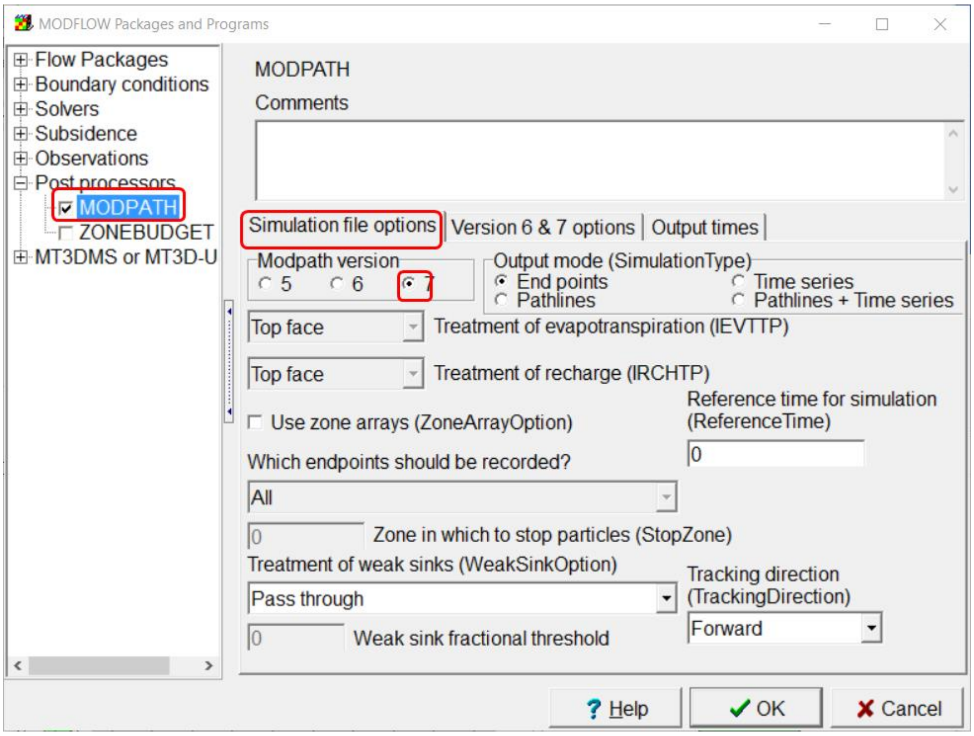


Figure Box 20-1 - Screen capture illustrating activating MODPATH 7 in the MODFLOW Packages and Programs dialog box.

4. On the Version 6 and 7 options tab, set STOPOPTION to stop the particles at the termination points ((Figure Box 20-2). Click *OK* to close the dialog box.

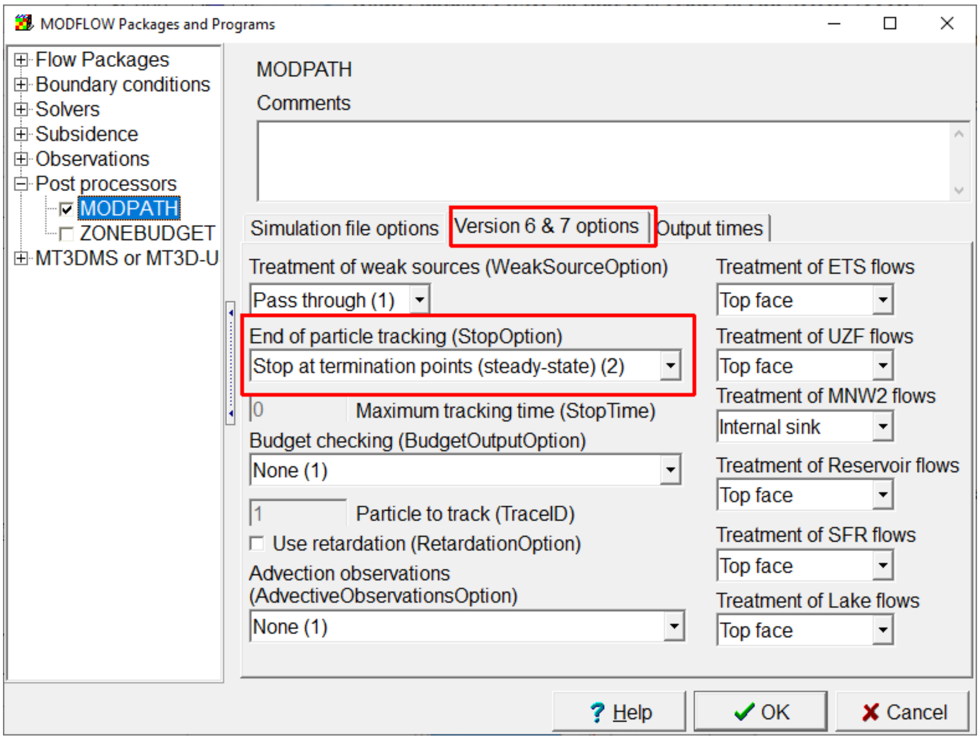


Figure Box 20-2 - Screen capture illustrating specifying options for MODPATH.

5. Make sure the installation location for MODPATH is specified correctly in the *Model|MODFLOW Program Locations* dialog box ((Figure Box 20-3).

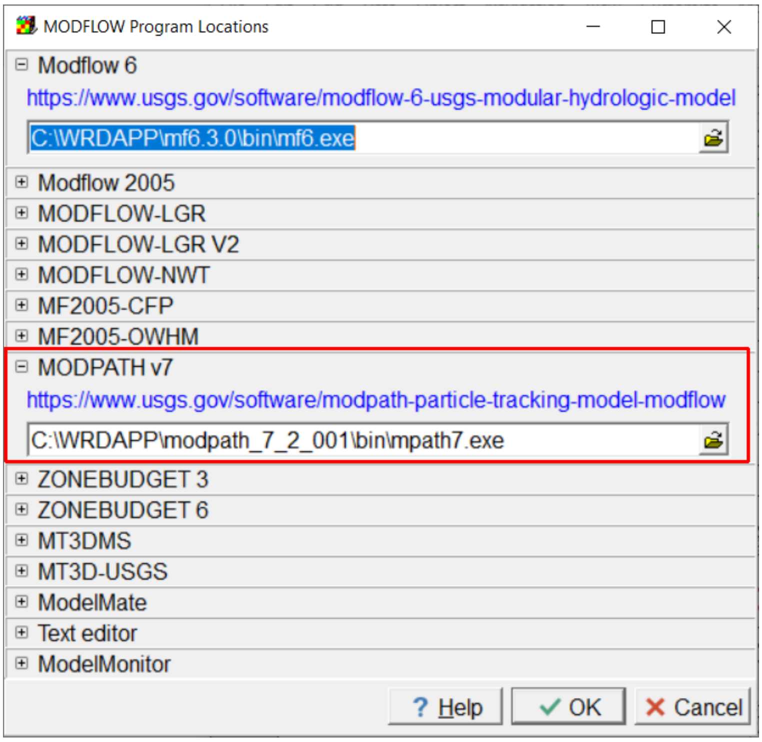


Figure Box 20-3 - Screen capture illustrating the MODPATH 7 installation location.

6. Select *Data|Edit Data Sets* and set the porosity for all layers to 0.3 ((Figure Box 20-4). Click on *Apply* to change the porosity formula.

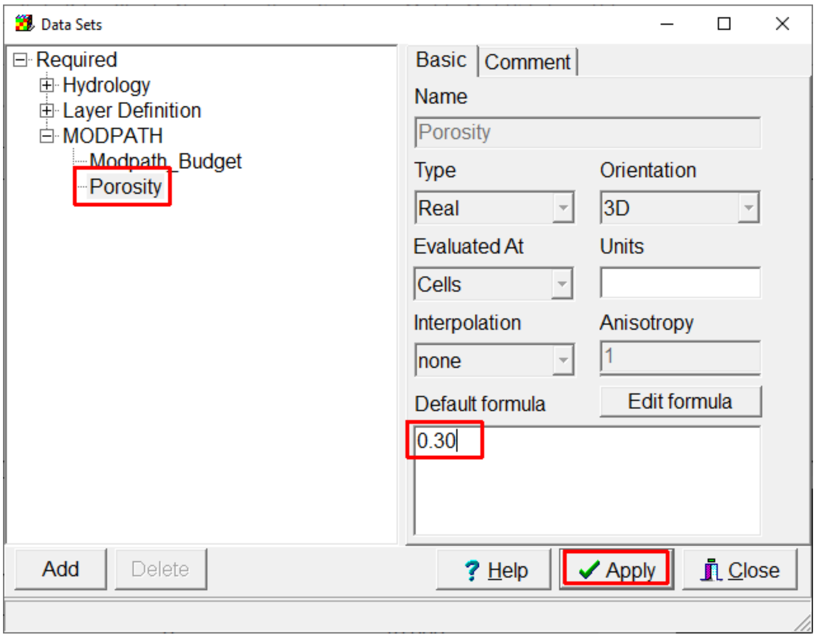


Figure Box 20-4 - Screen capture illustrating specifying porosity in the *Data|Edit Data Sets* dialog box.

7. The dataset should now be ready to run MODFLOW and MODPATH. However, there is one very important step required to make sure things are set up properly. When MODPATH is activated in ModelMuse, an auxiliary variable named IFACE is added to each entry in the river, drain, well, and recharge packages. IFACE is the integer boundary face flag that tells MODPATH how to assign those stresses to cell faces if they represent boundary flows. By default, ModelMuse will initially set IFACE equal to zero for all entries. A value of zero indicates that the stress component is distributed internally within the cell and not assigned to a cell face to represent a boundary flow. In this problem, the conceptual model of the river and drain is that they are shallow features that sit near the top of layer 1. To represent that, the IFACE values for the drain and river cells need to be assigned a value of 6, which indicates that MODPATH will assign those flow rates to the top faces of the cells. Recharge is also conceptualized as occurring at the surface, so it also needs to be assigned an IFACE value of 6. The IFACE value can be changed for each individual Objects in the Object Properties dialog box ((Figure Box 20-5). If the objects are not visible, display them with the *Object|Show or Hide Objects* dialog box. It can be helpful to hide the other objects. Select all three objects. This can be done on the top view of the model by clicking on them one at a time while holding down the *Shift* key on the keyboard. It can also be done in the Show or Hide Objects dialog box by selecting an object, right-clicking on it and selecting *Add to selection* from the popup menu. Once all three are selected, double

click on one of the selected objects on the top view of the model or click the *Edit selected objects* button in the Show or Hide Objects dialog box. Choose the MODFLOW Features tab and click on *MODPATH* (there should be a gray check next to DRN, RCH, and RIV), then in the upper portion under "Flow into boundary cell (IFACE)" click the button next to "Top (6)." Once those changes are made, the dataset is ready to be used for MODPATH simulation.

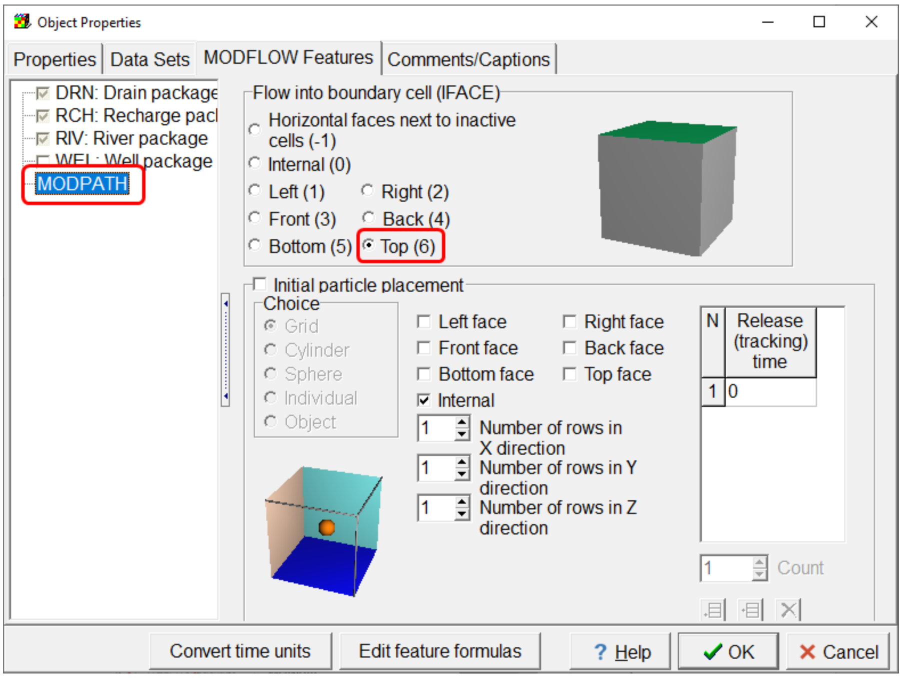


Figure Box 20-5 - Screen capture illustrating specification of the cell face through which the boundary flow occurs for MODPATH in the Object Properties dialog box.

8. Run MODFLOW again so that the budget output required by MODPATH includes data for all stress periods and the correct IFACE values before any MODPATH simulations can be made. Make sure "Export MODPATH input" is checked in the dialog box that appears after instructing ModelMuse to run the model. Save this version of the model. It will be the starting point for each of the following simulations.

Return to where text links to Box 20↥

Box 21 Step-by-Step Instructions for PSMP, Simulation 1

Exercise 7.1 – Forward Endpoint Simulation

These directions apply to creating the simulation in ModelMuse. If another GUI is used, the steps will be the same but the GUI features will differ. If it is not already open, open the version of PSMP saved after completing the steps described in Box 20. Save the file with a new file name such as PSMP_Endpoints. Make a forward-tracking endpoint simulation to delineate the source areas for the wells, drain, and river. Both forward and backward tracking can be modeled with MODPATH. The choice is specified in the MODFLOW Packages and Programs dialog box. This is also where you specify whether MODPATH produces endpoints, path lines, or time-series data.

1. Create a polygon or rectangle object that covers the entire top view of the model. In the Object Properties dialog box, select *MODFLOW Features* and check the box for Initial particle placement, then specify the particles on the top surface of the cell and place a 5×5 array of particles on face 6 (top face) of each cell in layer 1 (Figure Box 21-1).

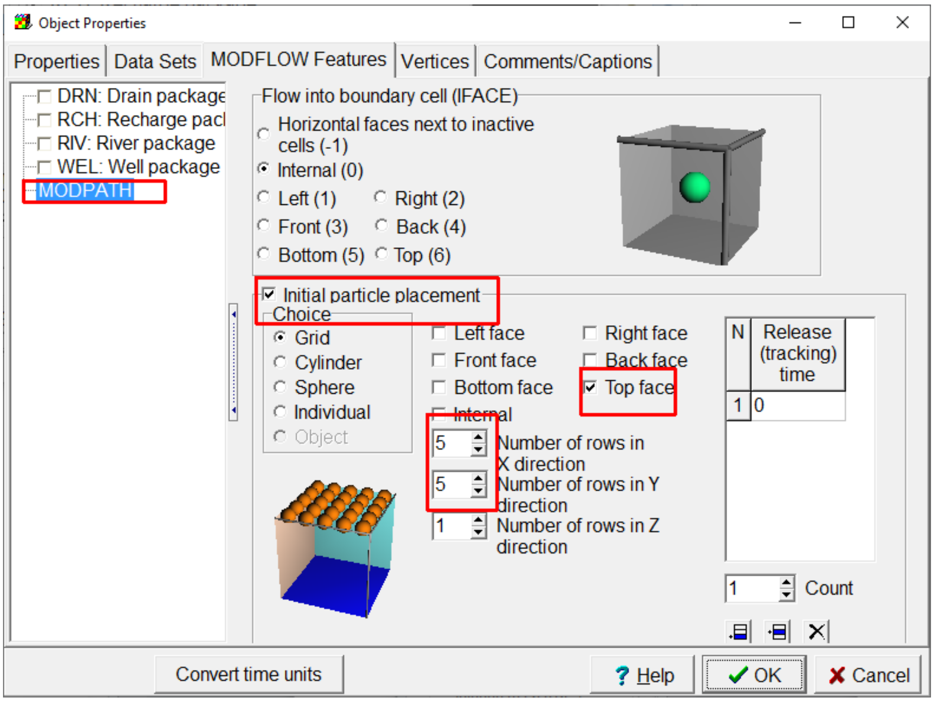


Figure Box 21-1 - Screen capture illustrating specifying the initial particle placement for MODPATH.

2. In the MODFLOW Packages and Programs dialog box, change the setting in MODPATH to use zone arrays (Figure Box 21-2). By default, forward tracking is already used.

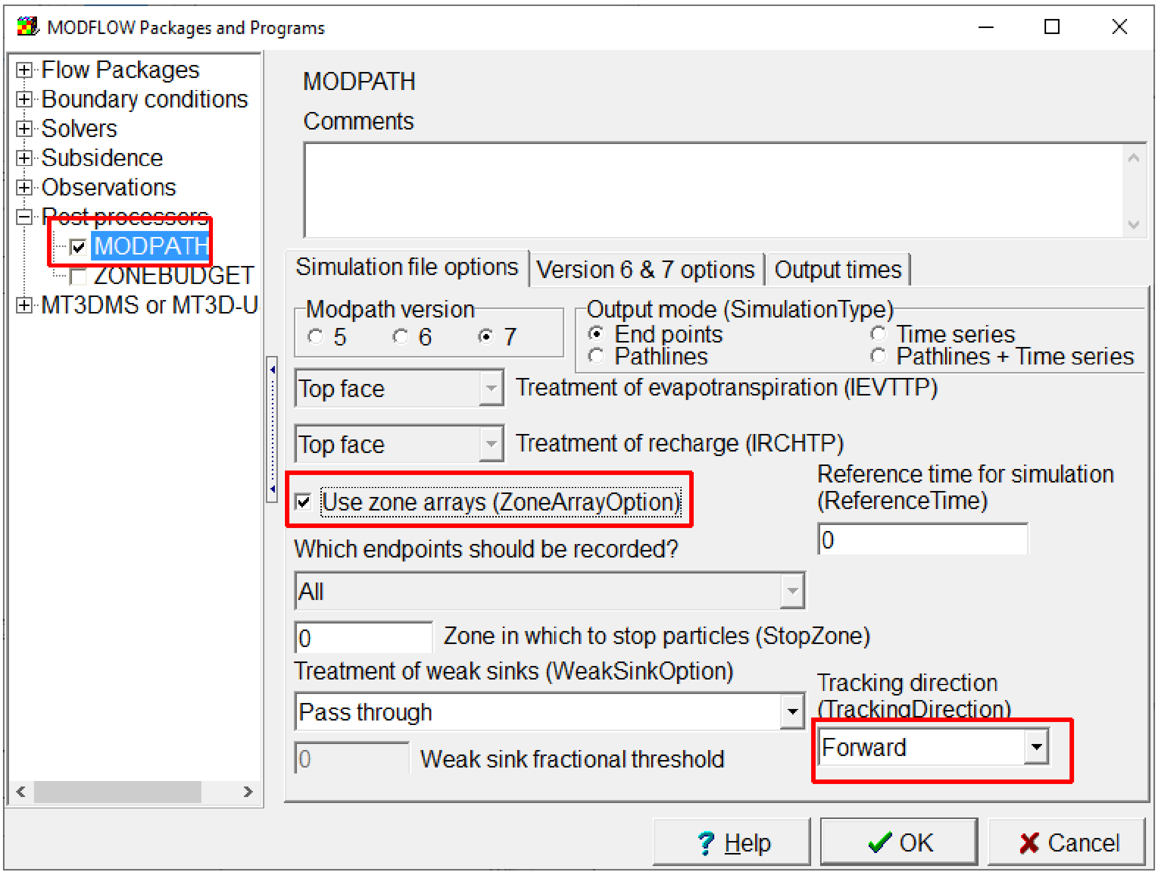


Figure Box 21-2 - Screen capture illustrating specifying options for MODPATH in the *Model|MODFLOW Packages and Programs* dialog box.

3. The default MODPATH zone number for the entire model is 1. Set the zone numbers of layer 1 and layer 3 so that the layer 1 cells where the river is located are zone 2, the cell of layer 1 that contains the shallow well is zone 3, the layer 1 cells containing the drain are zone 4, and the cell of layer 3 that contains the deep well is zone 5. The easiest way to do this is to select each object that specifies the boundary condition and use it to specify the MODFLOW Zone number on the Data Sets tab of the Object Properties dialog box (Figure Box 21-3). You can edit objects by double-clicking on them on the top view of the model or in the Show or Hide Objects dialog box. Note that you must unselect the previous selected Object or you may be changing the MODPATH zone numbers in the wrong locations.

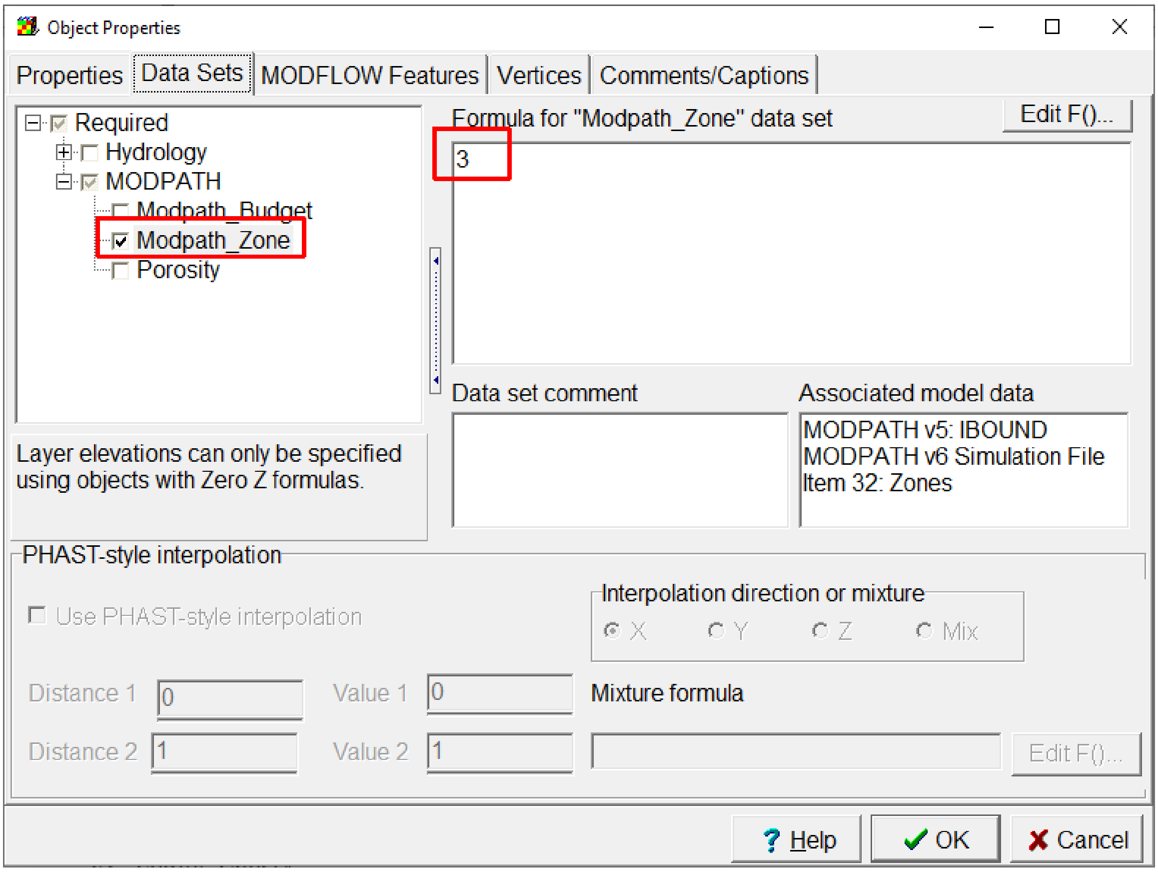


Figure Box 21-3 - Screen captures illustrating specifying the MODPATH zone numbers using the formula in an object.

4. MODPATH requires that the heads and flows be saved at every time step in binary format. You can make sure this is the case in the *Model|MODFLOW Output Control* dialog box (Figure Box 21-4).

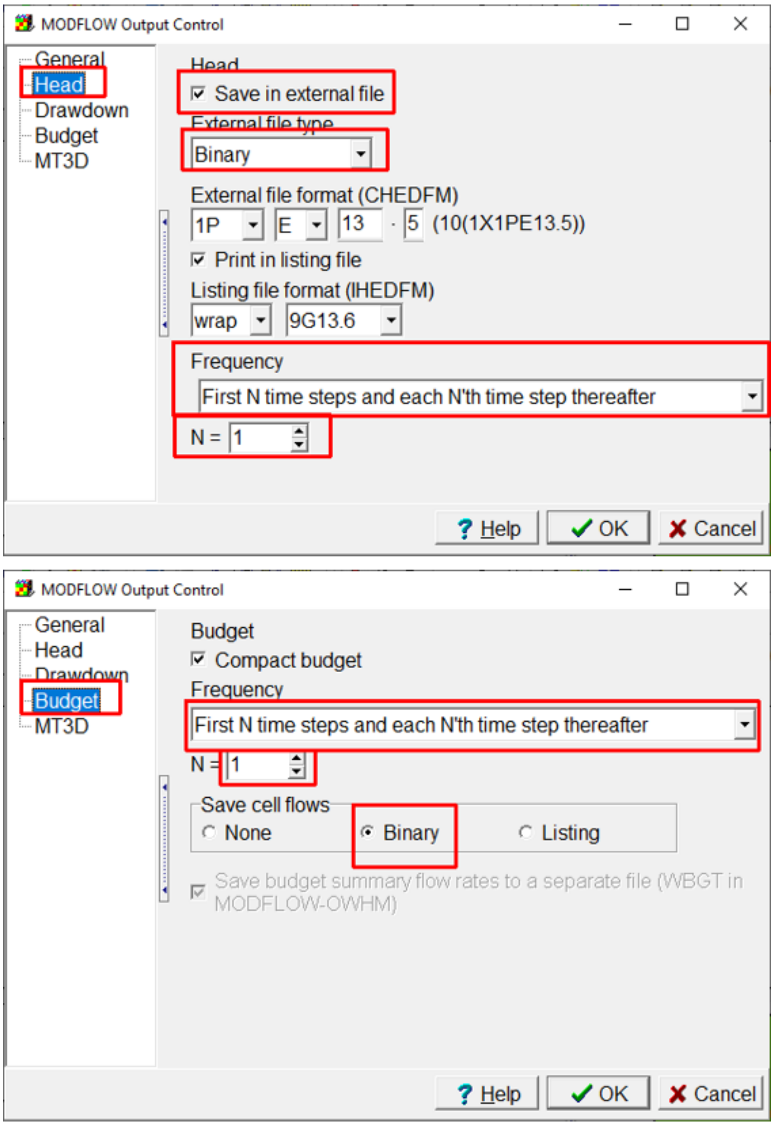


Figure Box 21-4 - Screen captures illustrating the required options for Output Control in MODFLOW when used with MODPATH.

5. MODPATH uses output files created by MODFLOW as part of its input. If you have not changed the name of the model, you do not need to run MODFLOW again to run MODPATH. Run MODPATH using the command *File|Export|MODPATH Input Files*. If you have changed the name of the model, you will need to run MODFLOW again before running MODPATH. When running MODFLOW, there is an option in the Save dialog box to run MODPATH after running MODFLOW. If you select that option, you can run both MODFLOW and MODPATH with a single command.
6. Plot the endpoints at their starting locations in ModelMuse with the particles colored by their final zone number. To plot the endpoints in ModelMuse, select *Data|Data Visualization* and select *MODPATH End Points*. Then select the endpoint file (Figure Box 21-5). The endpoint file will be in the same directory where you ran MODFLOW.

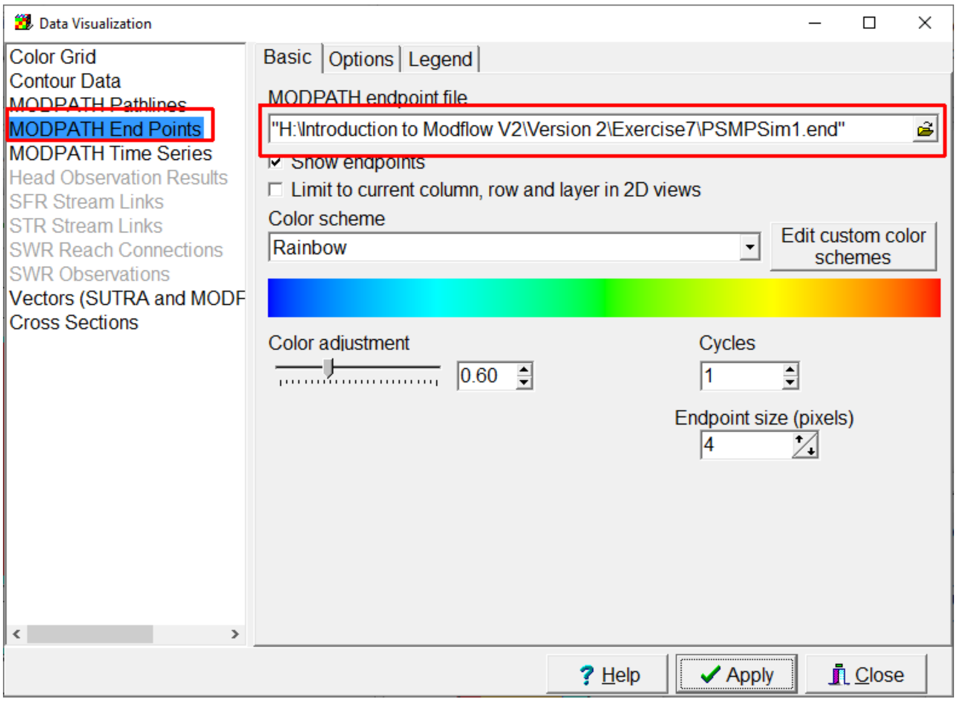


Figure Box 21-5 - Screen capture illustrating selecting an endpoint file for visualization in ModelMuse.

7. On the Options tab, set *Where to plot* to "Starting location" and choose to color by "Ending Zone" (Figure Box 21-6).

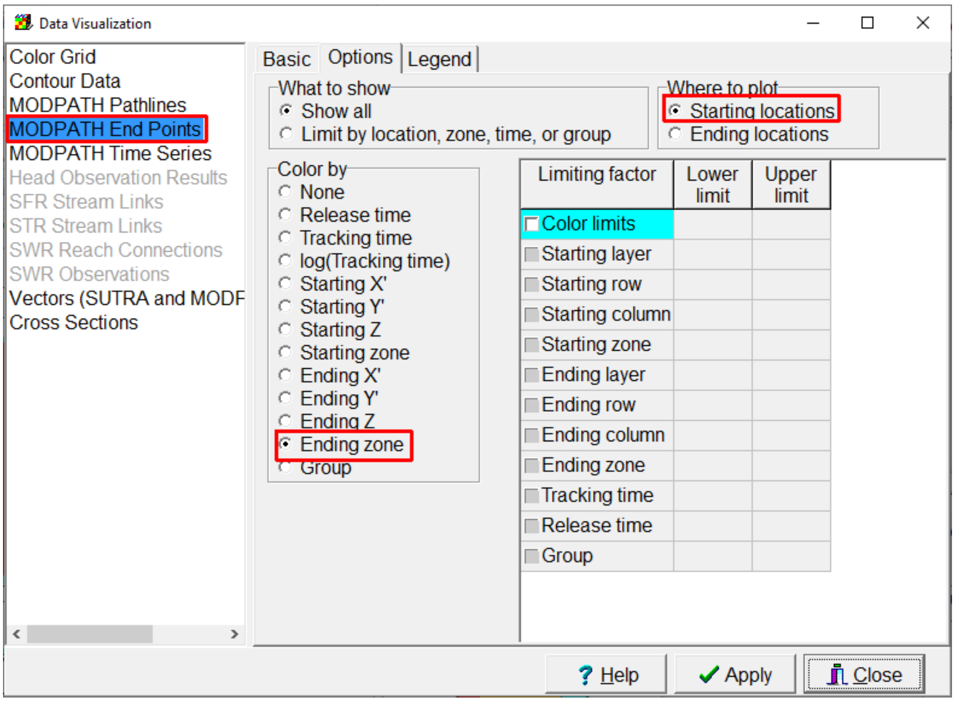


Figure Box 21-6 - Screen capture illustrating setting options for displaying endpoints.

8. How well do the MODPATH results compare with what you expected they would be when you completed PS2D1? Save the ModelMuse file.
9. Your results should resemble Figure Box 21-7.

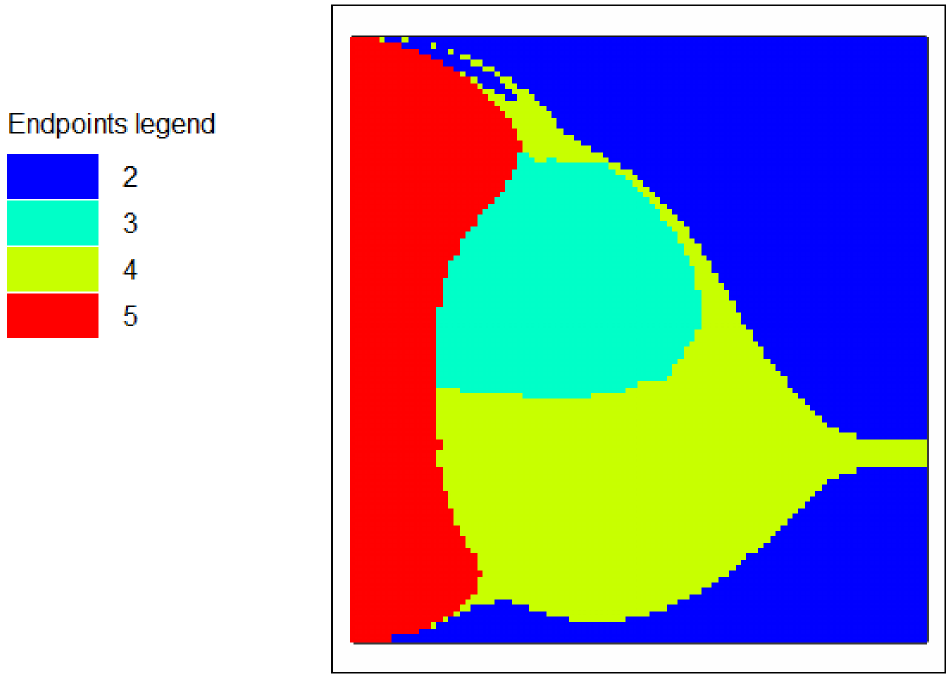


Figure Box 21-7 - Visualization of endpoint zones plotted at starting locations.

Return to where text links to Box 21

Box 22 Step-by-Step Instructions for PSMP, Simulation 2

Exercise 7.2 - Forward Pathline Simulation

Open the version of PSMP that was saved before starting the endpoint simulation and save it with a new file name such as PSMP_Pathlines. In this simulation, one particle will be placed on the top face of each cell in layer 1 (face 6) at X= 1750 and Y = 0 through -10,500 (column 4 and rows 1 through 21). After making a pathline simulation the results will be viewed in ModelMuse. Follow the steps below to create a new simulation file:

1. In the *Model|MODFLOW Packages and Programs* dialog box, set the simulation type to be Pathlines (Figure Box 22-1).

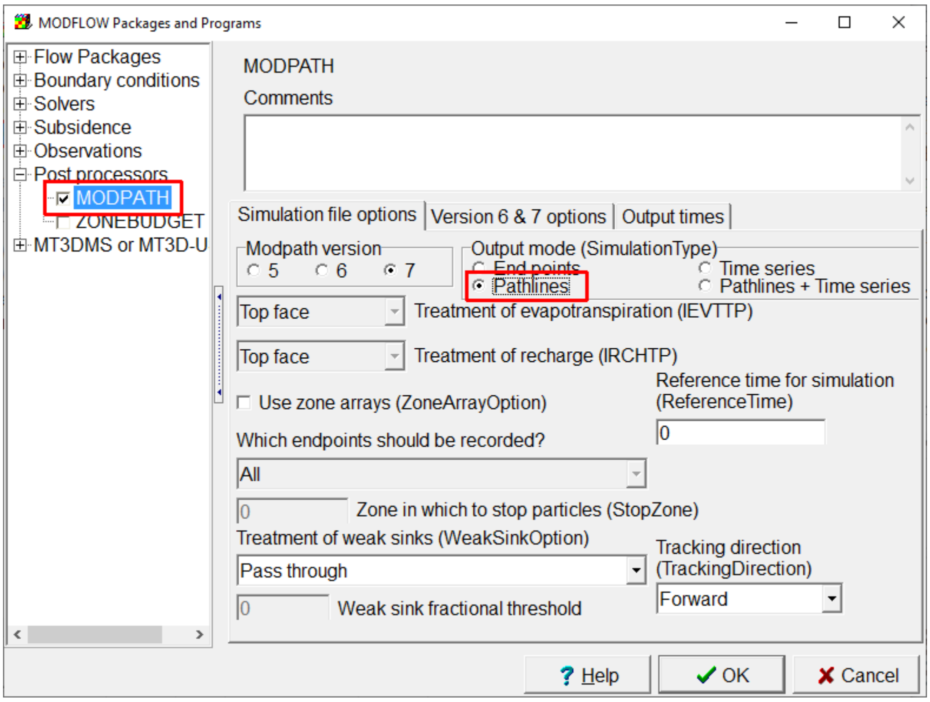


Figure Box 22-1 - Screen capture illustrating specifying the pathlines option in MODPATH.

2. The starting locations for particles should be in layer 1 at X= 1750 and Y = 0 through -10,500 (column 4 and rows 1 through 21). There should be 1 particle per cell placed in the center of the top surface of the cell. Create a line object in layer 1, column 4 going through all 21 rows (Figure Box 22-2).

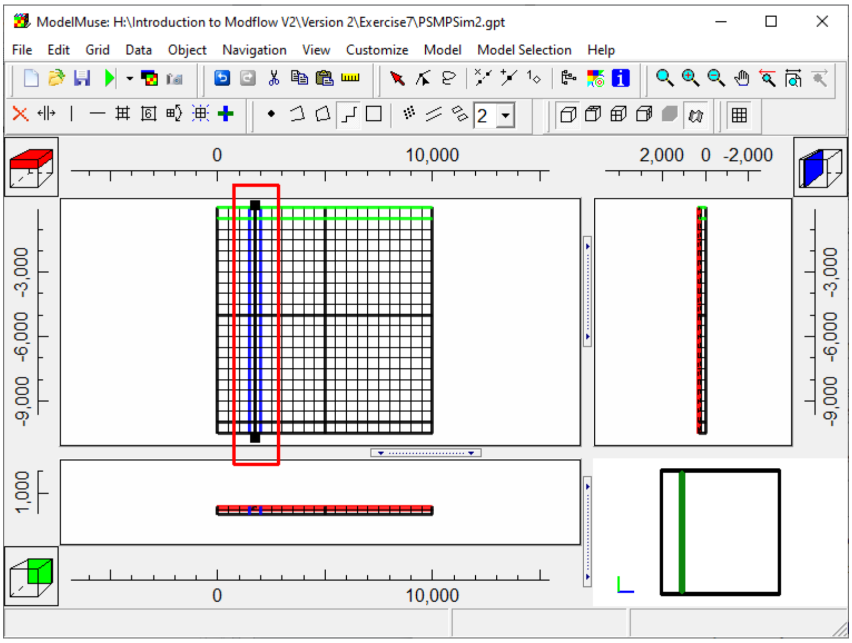


Figure Box 22-2 - Screen capture illustrating using an object to specify MODPATH particle starting locations.

3. In the Object Properties dialog box, set the MODPATH initial particle placement to be in the center of the top face of the cell (Figure Box 22-3).

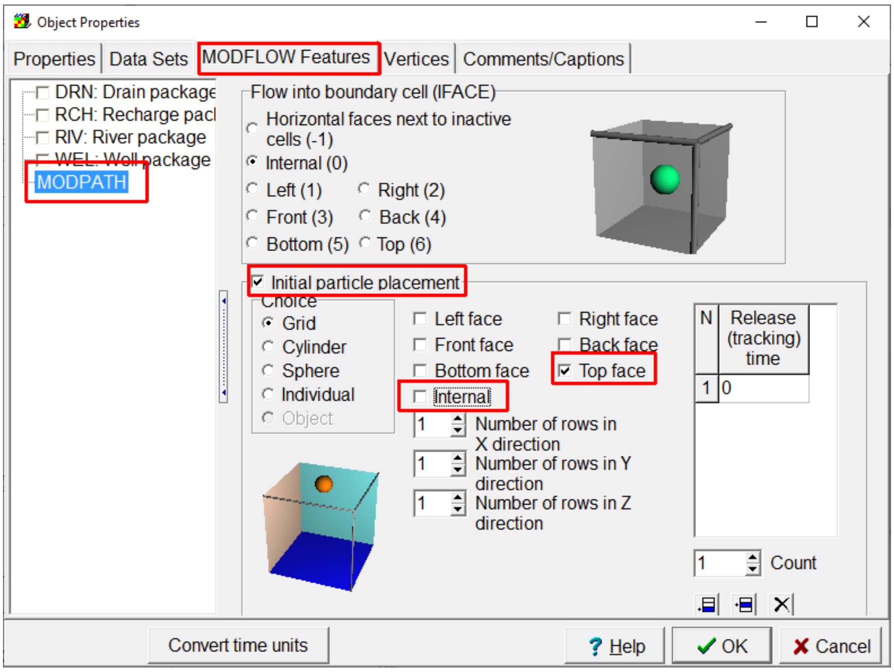


Figure Box 22-3 - Screen capture illustrating the initial particle placement.

4. All the data should now be set to run the pathline simulation. Run MODPATH. After MODPATH finishes, select *Data|Data Visualization* and select *MODPATH Pathlines*. Select the pathline file and uncheck the check box for limiting the visualization to the current column, row, and layer in 2D views. Then click *Apply* (Figure Box 22-4).

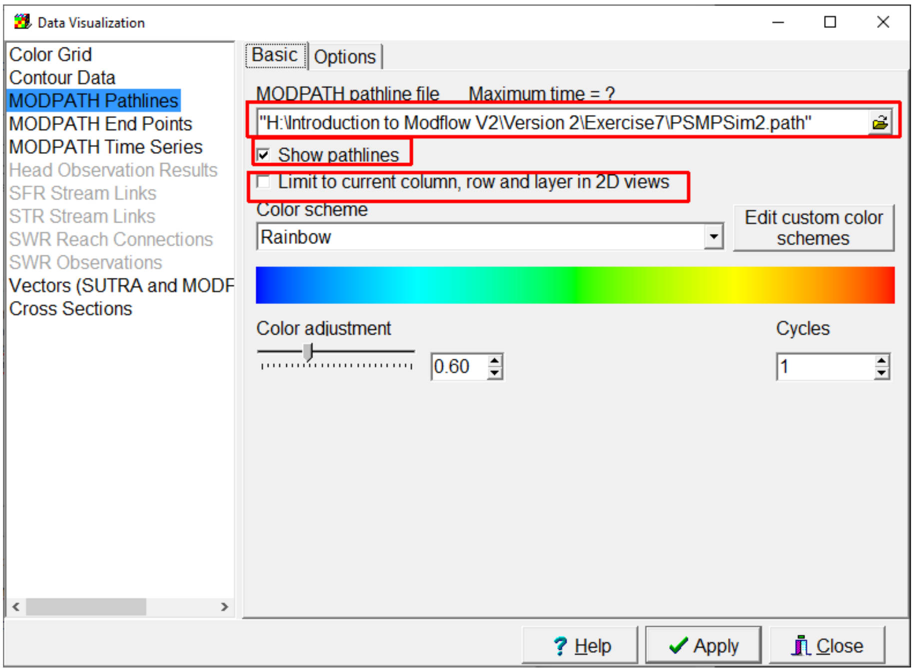


Figure Box 22-4 - Screen capture illustrating the selection of a pathline file.

7. If you wish to save results, save the ModelMuse file.
8. Your results may resemble Figure Box 22-5.

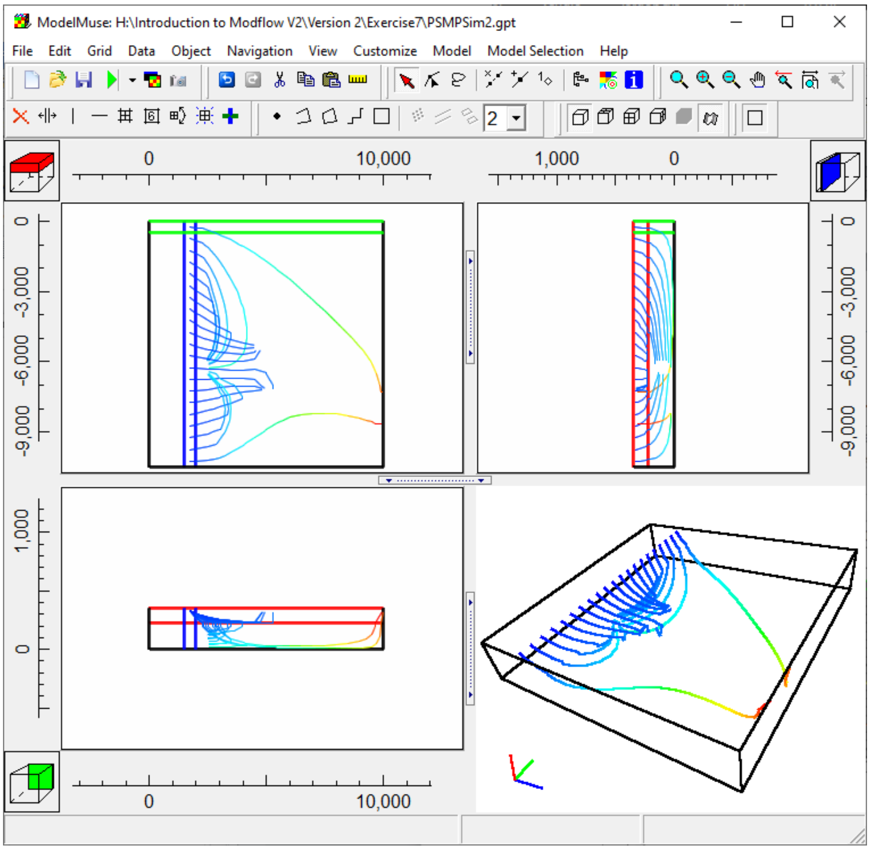


Figure Box 22-5 - Screen capture showing visualization of pathlines in top, front, side, and 3D views.

Return to where text links to Box 22

Box 23 Step-by-Step Instructions for PSMP, Simulation 3

Exercise 7.3 - Forward Timeseries Simulation

In this exercise a 5×5 array of particles is placed on the top face of layer 1 (face 6) for a block of cells in the range of row 8 through row 13 and column 4 through column 7 (coordinates = (1750±250, -4250±250) to (3250±250, -6250±250)). A time series simulation is then conducted in which particle locations are output to a time series file at a fixed time interval of 100 days for 300 time points. Steps 1 through 4 provide instructions for creating the simulation file.

1. Open the version of PSMP that was saved before starting the endpoint simulation and save it with a new file name such as PSMP_TiomeSeries.
2. Select *Model|MODFLOW Packages and Programs* and set the simulation type to Time Series. Define the time points for the timeseries simulation by specifying fixed time increments of 100 days and a total of 300 time points (Figure Box 23-1). Click *OK* to close the dialog box.

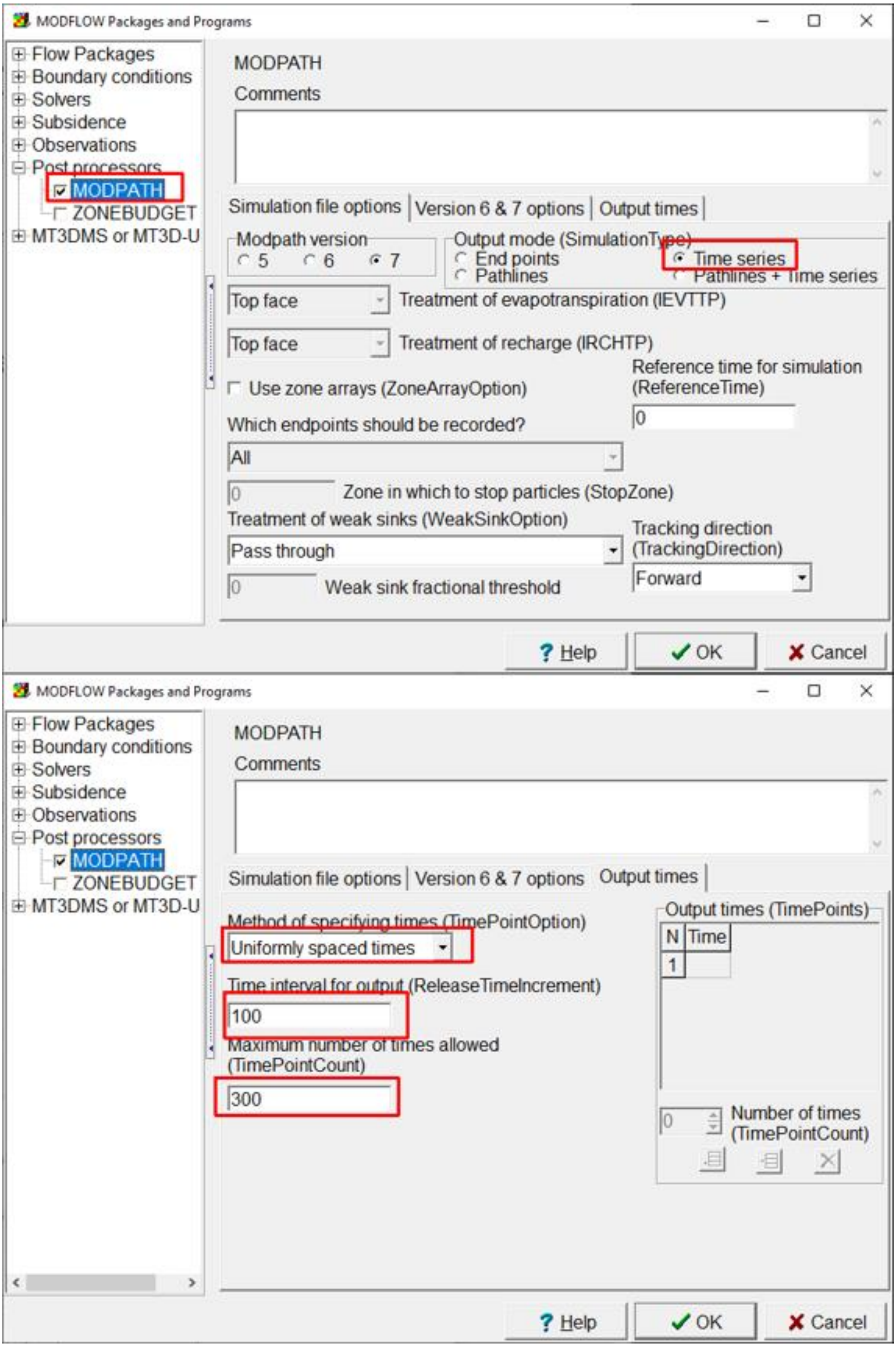


Figure Box 23-1 - Screen capture illustrating a) selection of the MODPATH time series option and b) related values.

3. Create a rectangle object on layer 1 covering the cells from row 8, column 4 to row 13, column 7. Use it to define a 5 by 5 array of particles for the top face of each cell (Figure Box 23-2). The particles will be assigned to any cell whose center is inside the object.

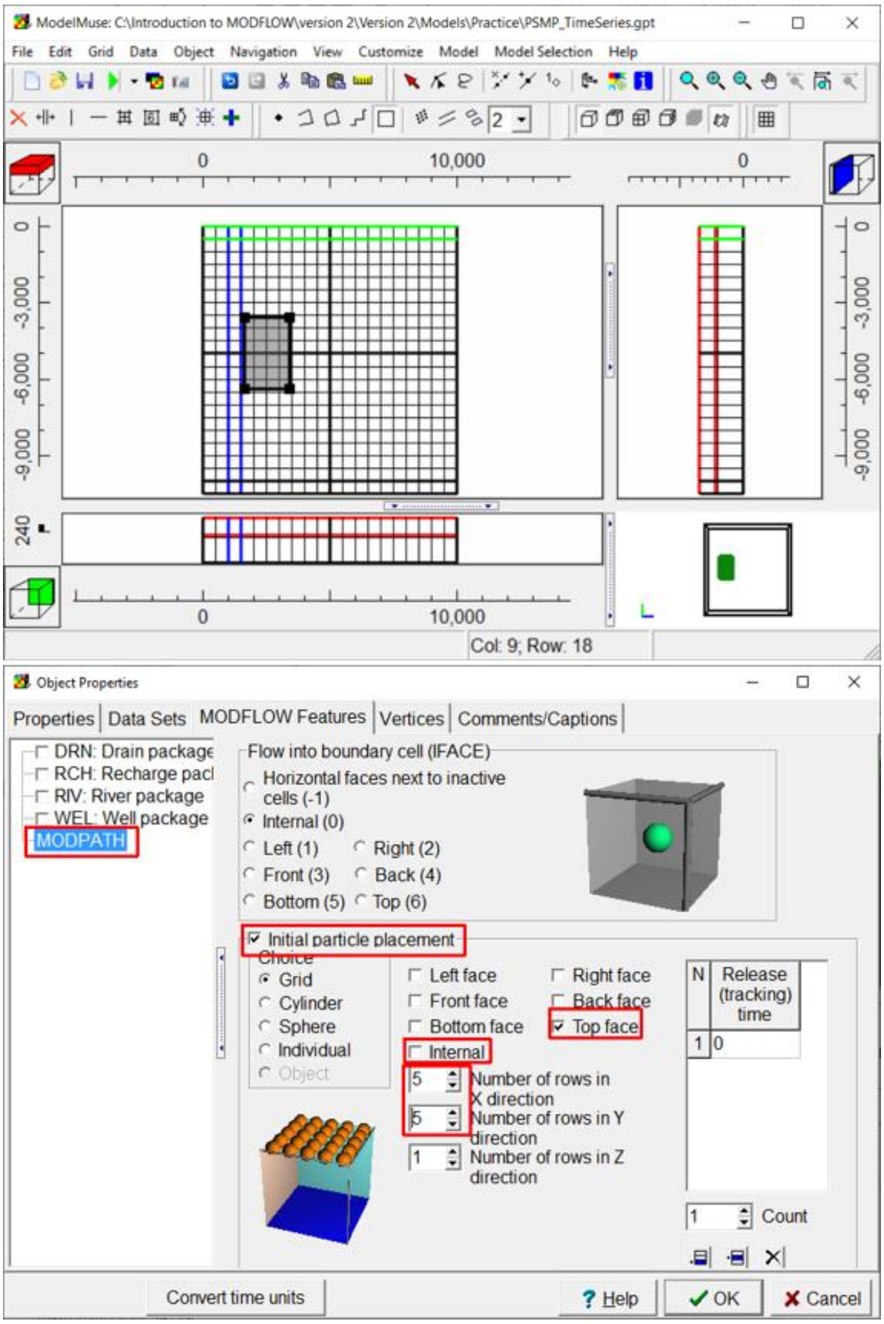


Figure Box 23-2 - Screen captures illustrating using an object to specify MODPATH particle starting locations.

4. Run MODPATH. After MODPATH finishes, select *Data|Data Visualization* and select *MODPATH Time Series* and the time series file. Uncheck the checkbox for limiting the view to the current column, row, and layer (Figure Box 23-3). If

desired, change options on the Options tab. Click on *Apply*. The plot can be animated by holding down the up-arrow button to the right of the "Time to plot" on the Basic tab in the Data Visualization window.

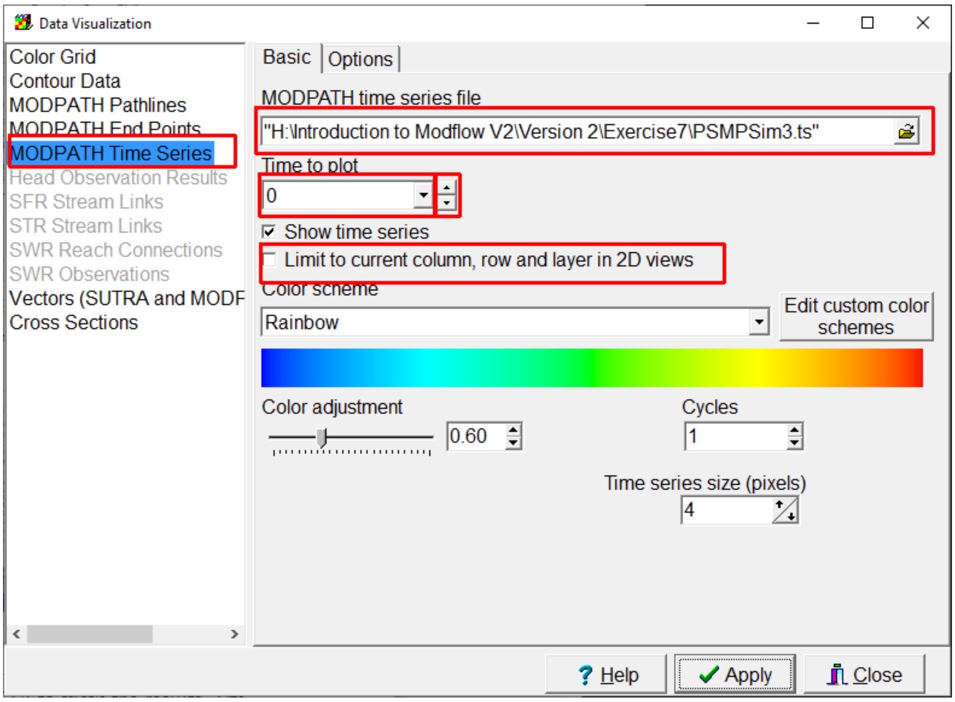


Figure Box 23-3 - Screen capture illustrating selection of MODPATH time series file for visualization.

5. Plots for the particles at selected times are shown in Figure Box 23-4. Once a particle exits the model through a boundary condition it is no longer tracked by MODPATH.
6. If you wish, save the ModelMuse file.

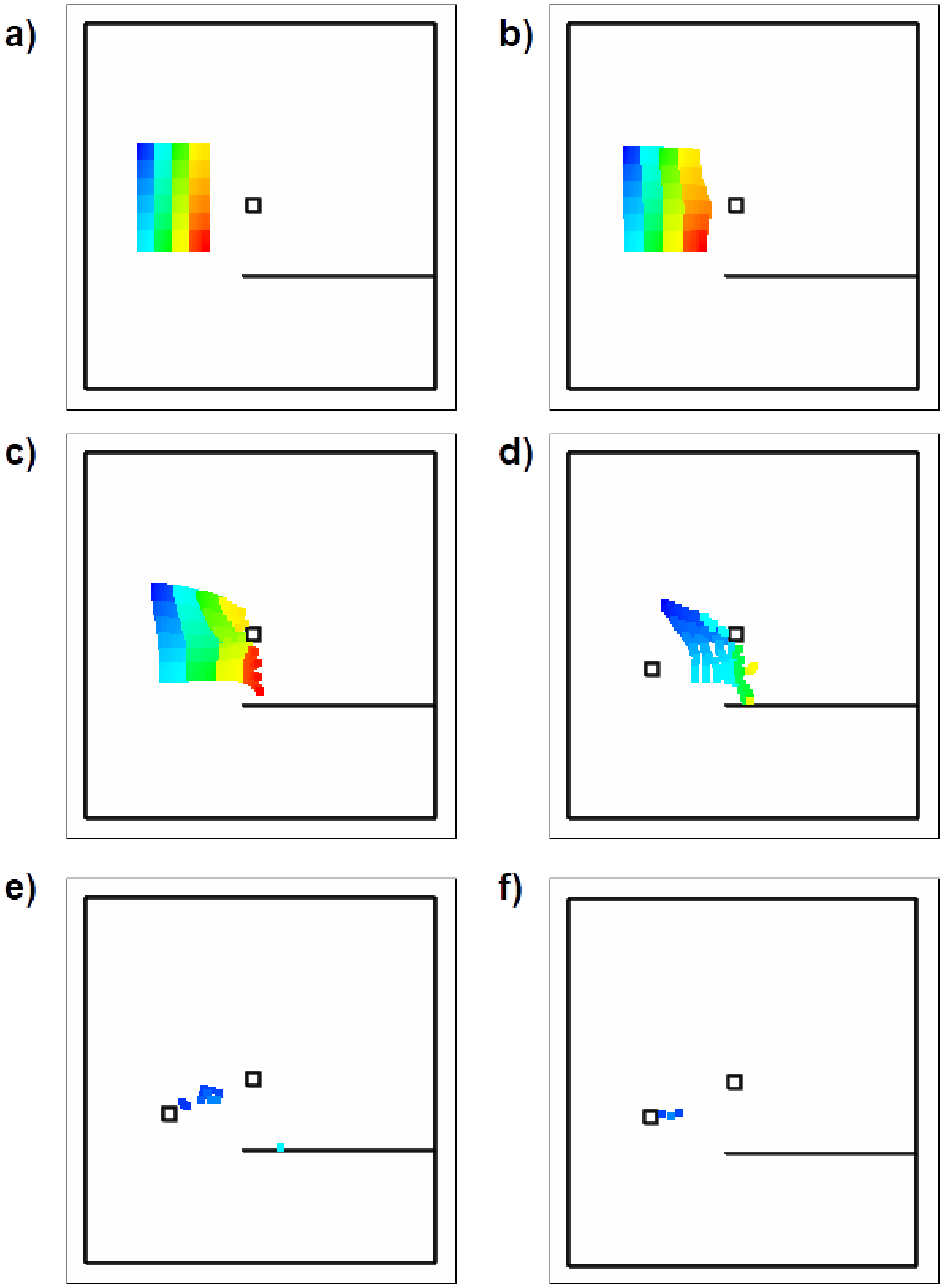


Figure Box 23-4 - Visualizations of MODPATH particles at time: a) 0 days; b) 1000 days; c) 4000 days; d) 9000 days; e) 20,000 days; f) 25,000 days. Animation of the particles reveals that particles reach the well in layer 1 at about 1600 days, the drain in about 4600 days, enter layer 2 after about 9200 days, enter layer 3 after about 11,700 days, and enter the well in layer 3 after about 17,300 days.

Return to where text links to Box 23

17 Exercise Solutions

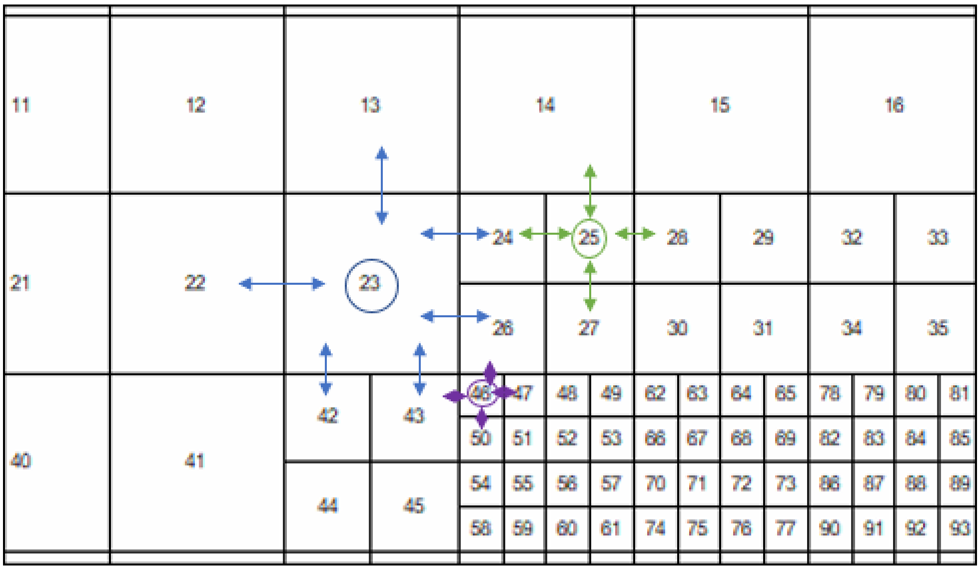
Solution Exercise 1

The option is determined by whether NOGRB is present as one of the OPTIONS. If NOGRB is missing, a binary grid file will be created. If it is present a binary grid file will not be created. In this case, NOGRB is not present so a binary grid file will be created.

In ModelMuse, you can specify this option in the *Model|MODFLOW Options* dialog box.

Return to Exercise 1↥

Solution Exercise 2



a. Flow from or to cell 23 can occur from cells 13, 22, 24, 26, 42, and 43.
b. Flow from or to cell 25 can occur from cells 14, 24, 27, and 28.
c. Flow from or to cell 46 can occur from cells 26, 43, 47, and 50.

Note that flow only occurs through cell faces not through cell corners.

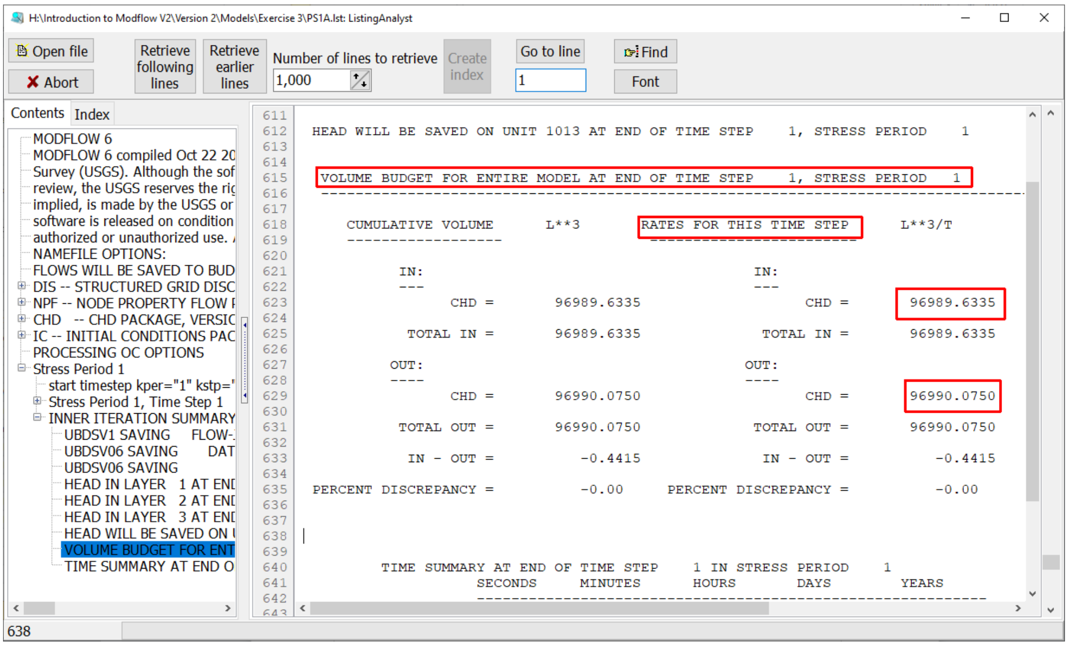
Return to Exercise 2↥

Solution Exercise 3

Solution to Exercise 3.1

As shown in the image below the volume budget at the end of the listing file shows the cumulative flow for the entire simulation on the left and the rates of flow for the time step on the right. All inflows are listed first and then outflows are listed. In this simple model there is only one type of boundary condition (CHD) and, because the modeler understands the hydraulics of the system, it is known that the entire CHD inflow is from the canal and the entire CHD outflow is to the river. The inflow along the entire 10,500 m of canal and the outflow to the entire 10,500 m of river (rounded to the nearest 1 m^3/day) are:

Inflow from canal = 96990 m^3/day   Outflow to river = 96990 m^3/day

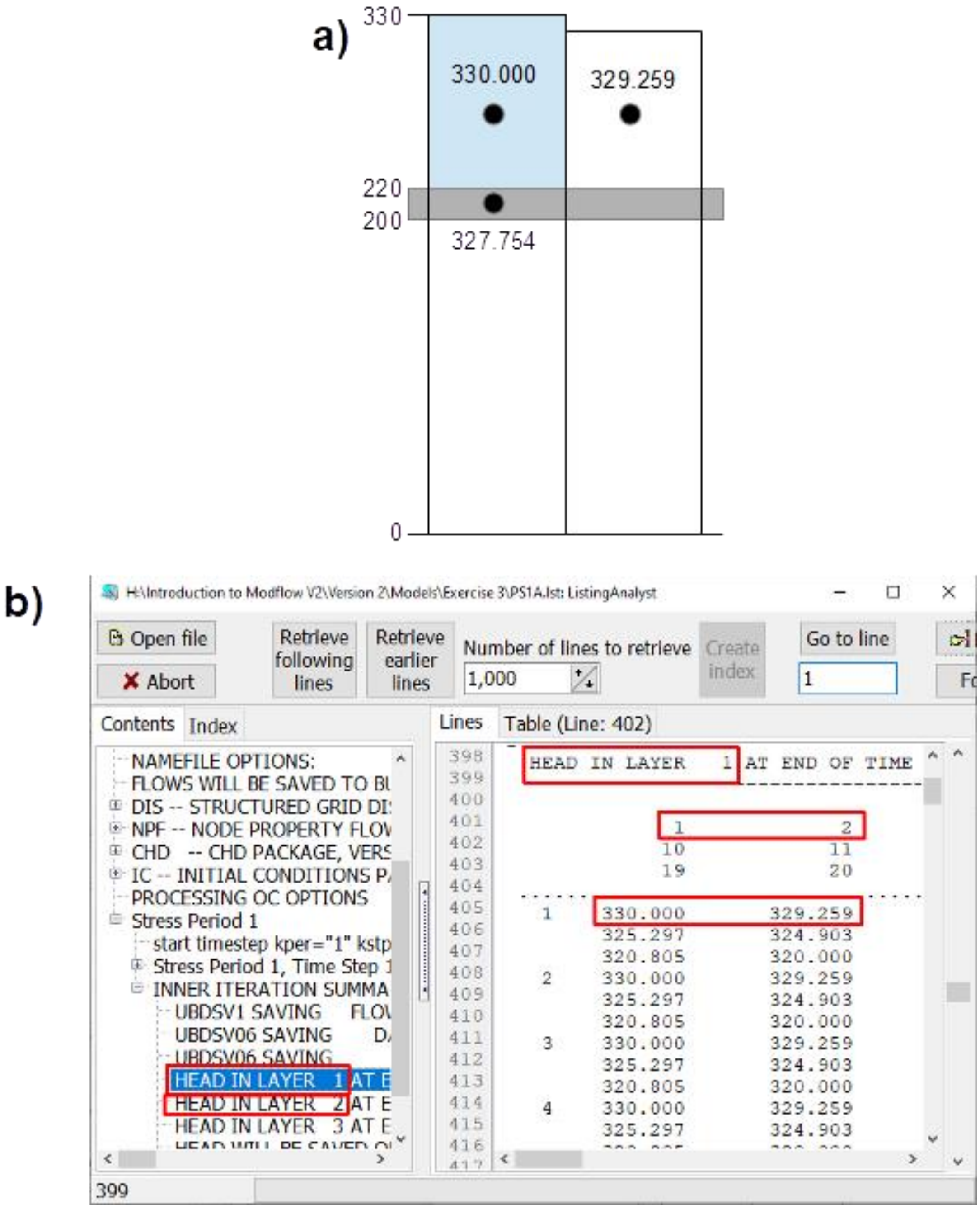


Screen capture of the volumetric budget section of the MODFLOW Listing file showing the flow rates through the CHD boundaries.

The slight difference in the listing file between the inflow and outflow results from minor numerical errors. If desired, these errors might be reduced by using smaller head closure values in the solver input file.

Water flows out of the canal into the groundwater system horizontally across the right face and downward across the bottom face. Both flows are into the groundwater system. The flows can be calculated by using Darcy's Law for flow from each cell, multiplying by 21 rows, then adding the results to get the total flow out of the canal.

To compute flow out the right face and the bottom face of one canal cell, first find the resulting heads as shown in the figure below. The contoured head values can also be viewed in ModelMuse by hovering the cursor on a cell and viewing the third panel of the status bar, but the listing file provides text output of the head calculated at each cell, that can be used to determine the value at the node location. The head in column 1 of layer 1 is the specified value of 330 m, in column 2 of layer 1 it is 329.259 and in row 1 of layer 2 it is 327.754. Simplify the calculation for flow out of the right face by using the average head between columns 1 and 2 ([330+329.259]/2 = 329.6295 m) and subtract the bottom elevation of layer 1 (220 m) to obtain the saturated thickness of layer 1 (109.6 m).



Illustrations containing data used to compute flow rates. a) Diagrammatic cross section of the first two columns with heads at cells used in computing flow rates. b) Screen capture of the MODFLOW listing file highlighting the location of the heads in layer 1 used in the flow calculations. Layer 2 heads are viewed by clicking on *HEAD IN LAYER 2* in the Contents menu.

The calculations are set up to give positive values for Q into the groundwater system.

$$Horizontal\ Flow = Q_H = \#\text{cells}\left(\text{K} \times \text{Length} \times \text{Thickness} \times \frac{\text{HeadDifference}}{\text{Distance}}\right)$$

$$Q_H = 21\left(50\frac{\text{m}}{\text{d}} \times 500\text{ m} \times 109.6\text{ m} \times \frac{330\text{ m} - 329.259\text{ m}}{500\text{ m}}\right) = 85{,}274.28\frac{\text{m}^3}{\text{d}}$$

$$Vertical\ Flow = Q_V = \#\text{cells}\left(\text{K} \times \text{Length} \times \text{Width} \times \frac{\text{HeadDifference}}{\text{Distance}}\right)$$

$$Q_V = 21\left(0.01\frac{\text{m}}{\text{d}} \times 500\text{ m} \times 500\text{ m} \times \frac{330\text{ m} - 327.754\text{ m}}{10\text{ m}}\right) = 11{,}791.5\frac{\text{m}^3}{\text{d}}$$

$$Q_{canal} = Q_H + Q_V = 97{,}065.78\frac{\text{m}^3}{\text{d}}$$

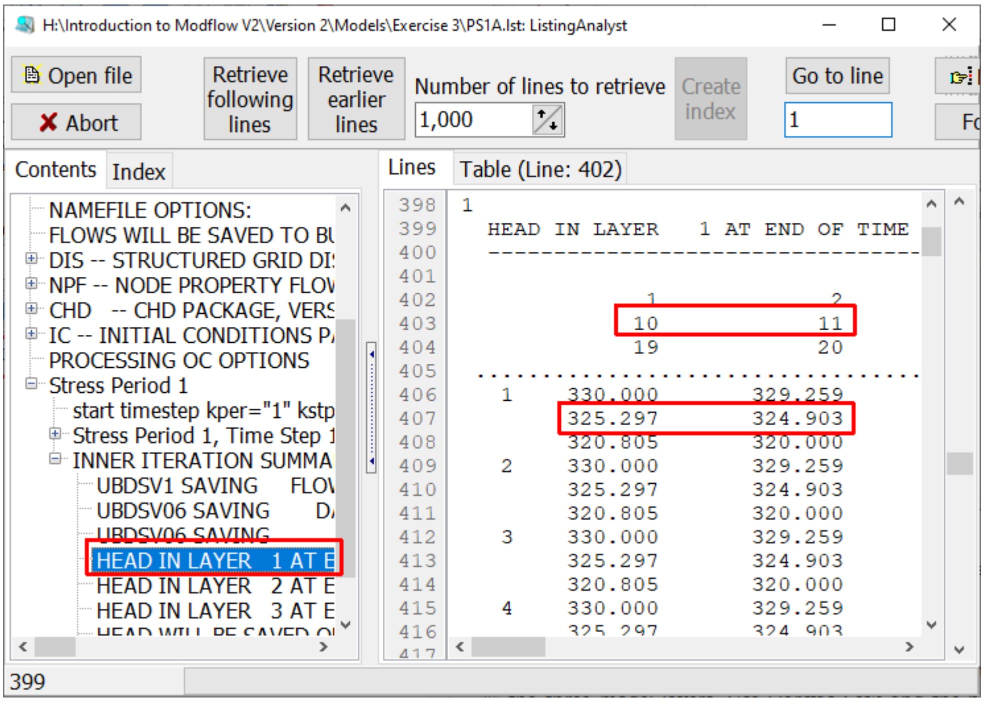
There are two reasons why the results of these calculations differ from those calculated by MODFLOW. First, the calculation of vertical flow assumes that the vertical hydraulic conductivity of the material between the centers of layers 1 and 2 has the hydraulic conductivity of layer 2. MODFLOW uses an inter-layer hydraulic conductivity based on the hydraulic conductivities of both layers 1 and 2. This accounts for most of the difference between MODFLOW's results and the hand calculations. Second, MODFLOW has less rounding error in its calculations.

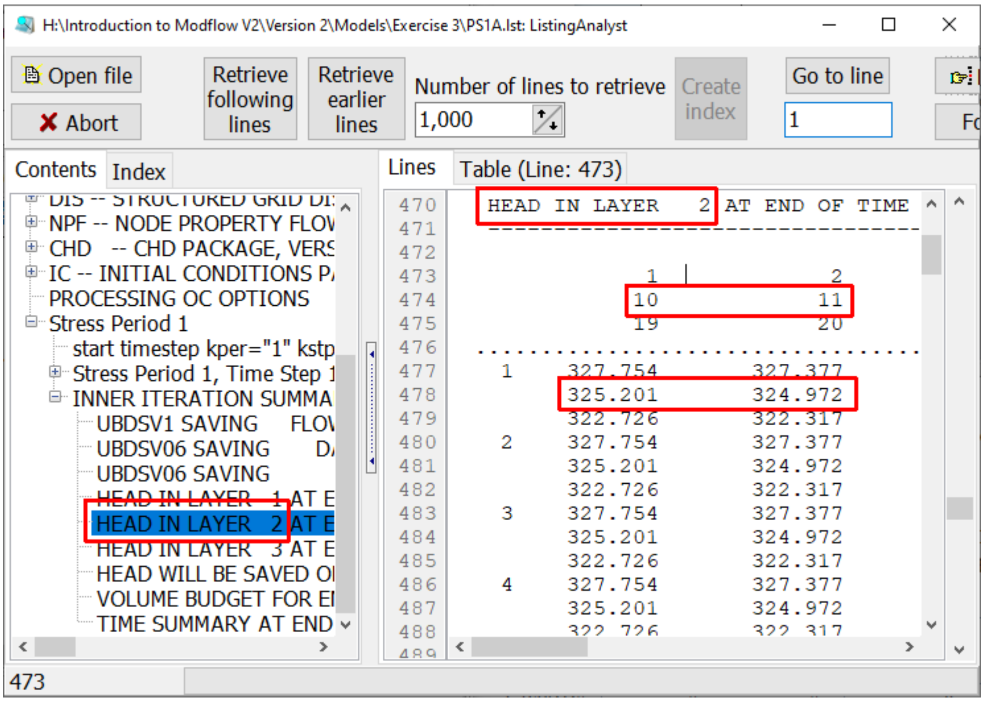
Specific discharge vectors can be displayed in ModelMuse by importing "Data-Spdis" from the cell-by-cell flow file and then selecting it as the vector source in *Data|Data Visualization* dialog box.

Return to Exercise 3.1↥

Solution to Exercise 3.2

The heads for each cell in layers 1 and 2 are compared using the listing file, which shows that the head in layer 1 is higher than layer 2 in column 10, while the head in layer 1 is less than the head in layer 2 in column 11, so flow directions change from downward to upward flow between columns 10 and 11 (as shown in the images below). This is also true for flow across the top of layer 3.





Screen captures of the parts of the MODFLOW listing file showing the heads in layers 1 and 2 used to determine the vertical direction of flow.

This can also be seen in the head contours of the cross section of the ModelMuse window.

Return to Exercise 3.2

Solution to Exercise 3.3

	h_{10}	h_{11}	Δh $(h_{11} - h_{10})$	b layer thickness	Q (all 21 rows)
Layer 1	325.297	324.903	-0.394	105.1	43,480
Layer 2	325.201	324.972	-0.229	20	1
Layer 3	325.105	325.041	-0.064	200	53,760

$$Q = [-KA/L\ \Delta h]21$$

$$\textit{Layer } 1: \quad Q = \left[-50\frac{m}{day} \times \frac{500m \times 105.1m}{500m}(-0.394m)\right]21 \approx \quad 43{,}480\ \frac{m^3}{day}$$

$$\textit{Layer } 2: \quad Q = \left[-0.01\frac{m}{day} \times \frac{500m \times 20m}{500m}(-0.229m)\right]21 \approx \quad 1\ \frac{m^3}{day}$$

$$\textit{Layer } 3: \quad Q = \left[-200\frac{m}{day} \times \frac{500m \times 200m}{500m}(-0.064m)\right]21 \approx \quad 53{,}760\ \frac{m^3}{day}$$

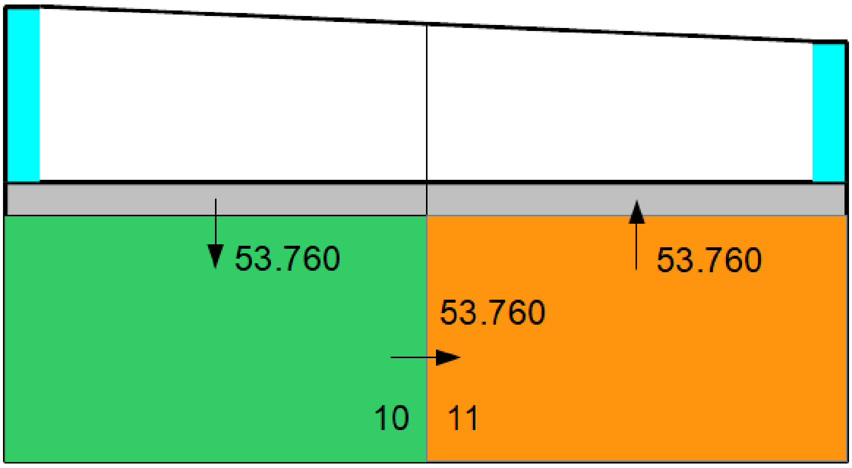
Total flow from column 10 to column 11 for all layers is the sum of the flow values in the last column of the table: Total flow between columns 10 and 11 = 97,241 m^3/d.

This value should be equal to the rate of flow into the groundwater flow system from the canal which is reported in the budget summary of the MODFLOW list file as 96,990. The slight difference (0.3%) between the values occurs because the hand calculations have less precision than the internal calculations made by MODFLOW.

Return to Exercise 3.3↥

Solution to Exercise 3.4

The easiest way to find the total downward rate of flow into layer 3 is to recognize that the flow rate calculated between columns 10 and 11 in layer 3 represents all the flow that makes it down into that layer (53,760 m^3/d). The geometry of the system is such that all that flow must enter across the top of layer 3 in columns 1 through 10. This exercise illustrates that there must be a water balance for any sub-region of the model as well as for the entire model. The total upward flow across the top of layer 3 in columns 11 - 20 must also equal 53,760 m^3/d based on the same reasoning. The flows are illustrated on the schematic cross section provided for the exercise as shown here.



The total downward flow could also be calculated by using Darcy's Law to compute the flow from layer 2 to layer 3 in columns 1 to 10 and summing the results. The same approach could be used to compute the upward flow in columns 11 - 20.

Return to Exercise 3.4↥

Solution to Exercise 3.5

ZONEBUDGET gives a different flow rate than was obtained from the hand calculation; 53,505 instead of 53,760. The difference in head used to calculate the flow between columns 10 and 11 had only two significant digits. ZONEBUDGET can calculate the flows more precisely because it can read the flows from the cell-by-cell flow package which were calculated with more precision than the hand calculations.

```
 VOLUME BUDGET FOR ZONE 3 AT END OF TIME STEP    1, STRESS PERIOD   1
 ---------------------------------------------------------------------------------------------

    CUMULATIVE VOLUME      L**3       RATES FOR THIS TIME STEP      L**3/T          PACKAGE/MODEL
    ------------------                 ------------------------                  ----------------

          IN:                                   IN:
          ---                                   ---
         DATA-SPDIS =          0.0000          DATA-SPDIS =          0.0000       NPF
                CHD =          0.0000                 CHD =          0.0000       CHD-1
             ZONE 1 =      53505.2120              ZONE 1 =      53505.2120
             ZONE 4 =          0.0000              ZONE 4 =          0.0000

           TOTAL IN =      53505.2120            TOTAL IN =      53505.2120

         OUT:                                  OUT:
         ----                                  ----
         DATA-SPDIS =          0.0000          DATA-SPDIS =          0.0000       NPF
                CHD =          0.0000                 CHD =          0.0000       CHD-1
             ZONE 1 =          0.0000              ZONE 1 =          0.0000
             ZONE 4 =      53505.1937              ZONE 4 =      53505.1937

          TOTAL OUT =      53505.1937           TOTAL OUT =      53505.1937

           IN - OUT =      1.8280E-02            IN - OUT =      1.8280E-02

PERCENT DISCREPANCY =          0.00     PERCENT DISCREPANCY =          0.00
```

Return to Exercise 3.5↥

Solution Exercise 4

Solution to Exercise 4.1

The flow through the system is doubled. The flow from the canal is the inflow from constant head in the MODFLOW budget, which has a value of 193,979 m^3/d (almost exactly double the value from PS1A of 96,990 m^3/d). However, doubling all the hydraulic properties has no effect on the head distribution.

This Exercise illustrates the classic problem that occurs when a modeler attempts to calibrate a model to head values without any knowledge of flow rates in the system. That is, in this case, head measurements do not provide any information about the magnitude of the hydraulic conductivity values because any multiple of the hydraulic conductivities produces the same head distribution. However, measuring the magnitude of groundwater discharge to the river can help identify the magnitude of the hydraulic conductivities, because in the model only certain hydraulic conductivity values will produce simulated discharge equal to the measured discharge.

Return to Exercise 4.1↑

Solution to Exercise 4.2

Flow rates are nearly the same throughout the system as in PS1A. The flow in layer 1 is slightly larger because the saturated thickness is a little larger. Heads throughout the system increased by about 1 foot. This Exercise illustrates that the primary driving force for this flow system is the head difference between the canal and the river, which remains the same as for PS1A (10 m). The calculated flow rates are shown in the table below -

	Head column 10	Head column 11	Head difference	K	Head gradient	Layer height	Cross-sectional area	Flow rate
Layer 1	326.297	325.902	0.395	50	0.00079	106.1	1114045	44005
Layer 3	326.104	326.040	0.064	200	0.00013	200	2100000	53760

The flow rates could also be calculated using ZoneBudget. It gives a flow rate of 44,008 m^3/d in layer 1 and 53562 m^3/d in layer 3. Rounding errors in the hand calculations are responsible for the differences between the hand calculations and the ZoneBudget results.

The water budget in the listing file indicates that the flow rate out of the CHD boundaries is 97573 m^3/d.

Return to Exercise 4.2↑

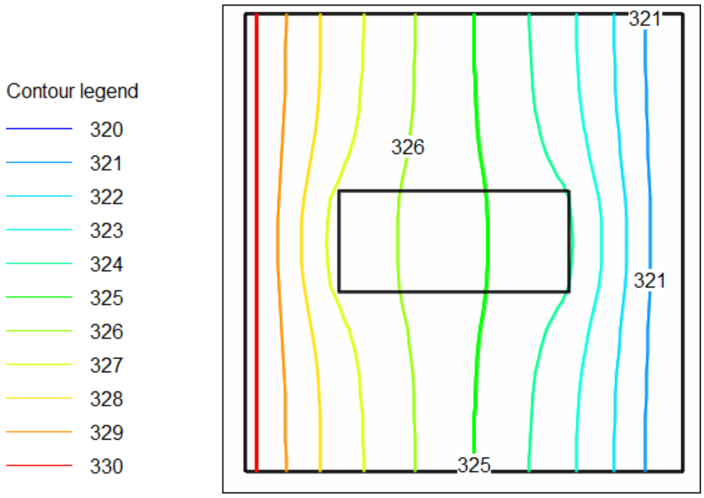
Solution to Exercise 4.3

The logic described in the solutions for Exercises 3.3 and 3.4 can be used to estimate flow from column 10 to 11 in layer 3. In the case of Exercise 3.3, the head difference between column 10 and 11 was 0.064 m, but for this case it is 0.001 m, so the volume of flow to layer 3 is greatly reduced (from 53,760 m^3/d to 840 m^3/d). Total flow through the system is reduced to 58,508 m^3/d when compared with approximately 96,990 m^3/d in PS1A. Almost all the flow occurs through layer 1. Because almost all the flow is in layer 1, the head gradient in layer 1 is closer to being uniform than it is in PS1B2.

Return to Exercise 4.3

Solution to Exercise 4.4

The head distribution is shown by going to *File|Import Results…* and selecting the file PS1B4.bhd then choosing *Contour Grid*. The contoured heads are shown in image included with this solution. The high conductivity zone in layer 1 results in smaller head gradients in the region of high conductivity. That, in turn, causes the head distribution to deform so that groundwater converges and moves into the upgradient part of the high conductivity zone and then diverges as it exits the zone at its downgradient end.



Visualization of heads in layer 1 in a model with a non-uniform hydraulic conductivity.

Return to Exercise 4.4

Solution Exercise 5

Problem Set 2: Introduction to Stress Packages

This problem set introduces the use of stress package features. The groundwater flow system is like the base case in problem set 1 (PS1A), except the canal on the left side has been removed and areal recharge has been added to layer 1 as the new source of water for the system. The recharge rate is 0.005 m/day.

The problem set consists of 4 major parts (A, B, C, and D). Part A introduces the use of the recharge package. Part A uses traditional data input to represent hydraulic properties and recharge. Parts B, C, and D demonstrate the river, drain, general-head boundary, and well packages.

Solution to Exercise 5.1

Recharge rate multiplied by model area =0.005 m/d × 500 m × 500 m × 21 × 20 = 525,000 m^3/d

Recharge from MODFLOW budget = 498,750 m^3/d

The difference occurs because the recharge applied to the constant head cells in column 20 is ignored by MODFLOW.

Recharge Volume = 0.005 m/d × 500 m × 500 m × 21 × 19 = 498,750 m^3/d

Return to Exercise 5.1⮥

Solution to Exercise 5.2

The budget in the MODFLOW listing file indicates the total flow rate into the river is 498,750 m^3/d as shown in the screen snap include with the is solution. This should match the inflow from recharge, and it does match.

```
 VOLUME BUDGET FOR ENTIRE MODEL AT END OF TIME STEP    1, STRESS PERIOD    1
 ------------------------------------------------------------------------------------------

    CUMULATIVE VOLUME      L**3       RATES FOR THIS TIME STEP      L**3/T        PACKAGE NAME
    ------------------                ------------------------                    ----------------

          IN:                                  IN:
          ---                                  ---
                RCH =      498750.0000               RCH =      498750.0000       RCH-1
                CHD =           0.0000               CHD =           0.0000       CHD-1

           TOTAL IN =      498750.0000          TOTAL IN =      498750.0000

         OUT:                                 OUT:
         ----                                 ----
                RCH =           0.0000               RCH =           0.0000       RCH-1
                CHD =      498749.3979               CHD =      498749.3979       CHD-1

          TOTAL OUT =      498749.3979         TOTAL OUT =      498749.3979

           IN - OUT =           0.6021          IN - OUT =           0.6021

PERCENT DISCREPANCY =           0.00     PERCENT DISCREPANCY =           0.00
```

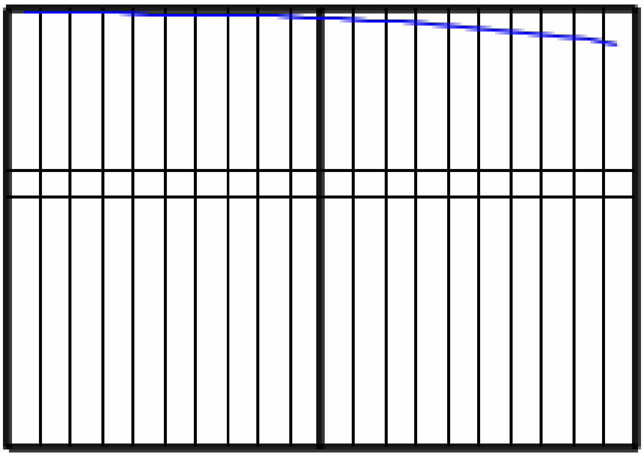
Screen capture of the MODFLOW listing file illustrating where the flow rate into the river (as represented by the CHD boundary) is listed.

Return to Exercise 5.2⮥

Solution to Exercise 5.3

The water table profile has more of a parabolic shape as shown in the profile with 20x vertical exaggeration in the image included with this solution. Vertical exaggeration can be set in ModelMuse using *View|Vertical Exaggeration…*.

The head profile is relatively flat near the left boundary and is progressively steeper toward the river on the right. The difference reflects the fact that uniform areal recharge continually adds flow to the system starting at the left boundary and moving toward the river. The water table gradient must increase toward the river to accommodate the increased flow.



Visualization of the water table.

Return to Exercise 5.3

Solution to Exercise 5.4

The change from downward to upward flow across the bottom of layer 1 can be found by comparing the heads in layers 1 and 2 in the listing file. Downward flow occurs when the head in layer 1 is greater than layer 2 and upward flow occurs where the head in layer 1 is less than the head in layer 2. The change occurs between columns 13 and 14.

Return to Exercise 5.4

Solution to Exercise 5.5

The total volumetric rate of flow between columns 13 and 14 in each layer is shown below. The procedure for calculating these flows is shown in detail in the Solution to Exercise 3.3.

	h_{left}	h_{right}	Δh $h_{right} - h_{left}$	b Thickness	Q (All 21 rows)
Layer 1	338.750	337.185	-1.565	117.9675	193,850
Layer 2	338.606	337.736	-0.87	20	4
Layer 3	338.461	338.286	-0.176	200	147,000

Return to Exercise 5.5

Solution to Exercise 5.6

The procedure presented in Exercise 3.1 that outlines calculation of flow (Q) across a model cell face is used to calculate the flow across each internal face of cell (1,1,10) and (1,1,20). The internal faces of cell (1,1,10) with flow are Q_{East}, Q_{Bot}, and Q_{West}. There is no flow across the north and south faces because heads are identical in all rows of any column of the model. The top, external, face of cell (1,1,10) receives recharge of 0.005 m/d and its area is 500 m by 500 m, so the volumetric flow across the top face (Q_{Top}) is [0.005 m/d × 500 m × 500 m] = 1250 m^3/d.

The top, external, face of cell (1,1,20) is outflow to the river and the east face is a no-flow (Q=0) boundary. The internal faces with flow are Q_{West} and Q_{Bot}. Flow to the river can be calculated as the sum of the other flows to the cell, or it can be determined by dividing the total flow out to the river CHD boundary by 21 (for the number of cells). The flows and resulting budget (or mass balance) are presented here for each cell. All flows are given in m^3/d. First the method used to calculate Q_{Bot} for cell (1,1,10). The head difference between layers 1 and 2 is much less in column 10 than in column 20 so the results of the hand calculations are more greatly affected by rounding error. For that reason, it is better to use column 20 for the detailed explanation. Thus, the method for cell (1,1,20) is described in more detail after presenting the case for cell (1,1,10).

Cell (1,1,10)

$$Q_{Out} = Q_{East} + Q_{Bot}$$

$$Q_{In} = Q_{West} + Q_{Top}$$

$$Q_{East} = 6001\ \frac{m^3}{d}$$

$$Q_{Bot} = 408\ \frac{m^3}{d}$$

$$Q_{West} = 5157\ \frac{m^3}{d}$$

$$Q_{Top} = 0.005\frac{m}{d} \times 500\text{m} \times 500\text{ m} = 1250\ \frac{m^3}{d}$$

$$Q_{Out} = 6001\ \frac{m^3}{d} + 408\ \frac{m^3}{d} = 6409\frac{m^3}{d}$$

$$Q_{In} = 5157\frac{m^3}{d} + 1250\ \frac{m^3}{d} = 6407\frac{m^3}{d}$$

$$Mass\ Balance = Q_{Out} - Q_{In} = 6409\ \frac{m^3}{d} - 6407\ \frac{m^3}{d} = 2\frac{m^3}{d}$$

$$\text{Balance Error=}\frac{|Difference\ between\ Out\ and\ In|}{Average\ of\ Out{+}In} = \frac{2\frac{m^3}{d}}{\frac{6409\frac{m^3}{d}+6407\frac{m^3}{d}}{2}} = 0.0003\ \ldots\ \text{or } 0.03\%$$

Cell (1,1,20)

Calculation of flow to the bottom of cell (1,1,20) is more involved than has been presented previously in this book because the adjacent cells have different hydraulic conductivity, thus the thickness-weighted harmonic mean conductivity, also called the equivalent conductivity, must be used in Darcy's law. The thickness-weighted harmonic mean is calculated as the quotient of the sum of the thicknesses and the sum of the ratios of thickness to hydraulic conductivity, as follows.

$$K_{harm} = \text{thickness-weighted harmonic mean}$$

The thicknesses and hydraulic conductivity values that contribute to the equivalent conductivity between layer 1 and 2 in column 20 are shown in the image below and the calculation results in a K_{harm} of 0.0597 m/d.

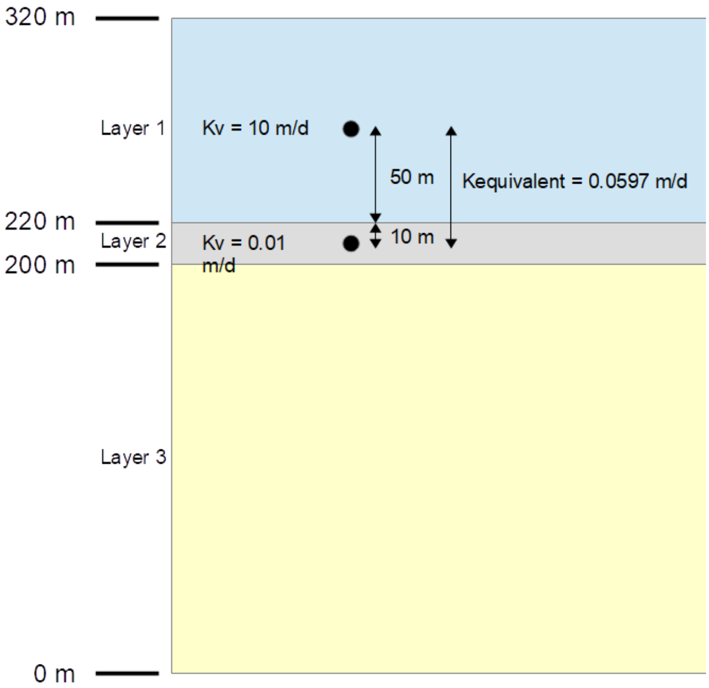


Illustration of thickness and hydraulic conductivity that contribute to the equivalent hydraulic conductivity between layer 1 and 2 in column 20.

$$K_{harm} = \frac{\sum thickness}{\sum \frac{thickness}{K}} = \frac{50\ m + 10\ m}{\frac{50\ m}{10\ m} + \frac{10\ m}{0.01\ m}} = 0.0597\ \text{m/d}$$

$$Q_{Bot} = K\frac{\Delta h}{\Delta l}A = 0.0597\ \frac{m}{d}\ \frac{328.769\ m - 320\ m}{60\ m}\ 500m\ 500m = 2181.289\ \frac{m^3}{d}$$

Continuing with the inflow and outflow of cell(1,1,20):

$$Q_{Out} = Q_{\text{River}}$$

$$Q_{In} = Q_{West} + Q_{Bot}$$

$$Q_{River} = \frac{\text{River CHD Outflow}}{\text{\# River cells}} = \frac{498{,}750\frac{m^3}{d}}{21} = 23{,}750\frac{m^3}{d}$$

$$Q_{West} = 21{,}582\ \frac{m^3}{d}$$

$$Q_{Bot} = 2{,}181\frac{m^3}{d}$$

$$Q_{Out} = 23{,}750\ \frac{m^3}{d}$$

$$Q_{In} = 21{,}582\ \frac{m^3}{d} + 2{,}181\ \frac{m^3}{d} = 23{,}763\ \frac{m^3}{d}$$

$$Mass\ Balance = Q_{Out} - Q_{In} = 23{,}750\ \frac{m^3}{d} - 23{,}763\ \frac{m^3}{d} = -13\frac{m^3}{d}$$

$$\text{Balance Error}= \frac{|Difference\ between\ Out\ and\ In|}{Average\ of\ Out{+}In} = \frac{13\frac{m^3}{d}}{\frac{23750\frac{m^3}{d}+23763\frac{m^3}{d}}{2}} = 0.0005\ \ldots\ \text{or}\ 0.05\%$$

Return to Exercise 5.6

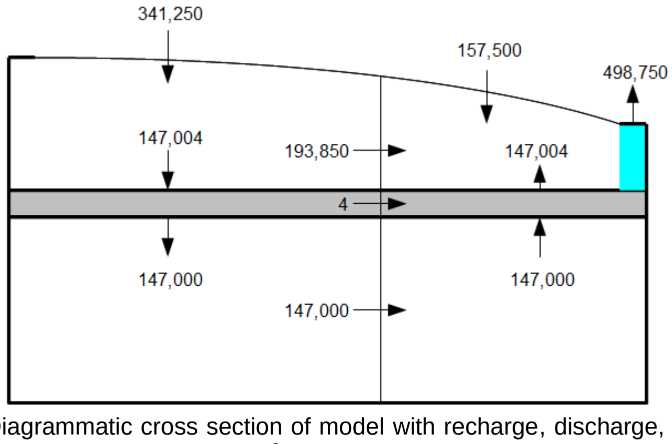
Solution to Exercise 5.7

The recharge and river discharge remain unchanged because they are not affected by changes in hydraulic conductivity. Head gradients throughout the system are smaller because larger hydraulic conductivities require smaller head gradients to move the same volumetric flow rate. The water table elevation in column 1 is lower in the PS2A1 model, declining from 346.044 m in run PS2A to 333.456 m. The proportion of the total flow that makes it into layer 3 is less (76,440 m^3/d) than in run PS2A (147,000 m^3/d as calculated in Exercise 5.5).

Return to Exercise 5.7

Solution to Exercise 5.8

The values are calculated using procedures described in previous exercise solutions and labeled in m^3/d in the image below.



Diagrammatic cross section of model with recharge, discharge, and internal flow rates in m^3/d.

Return to Exercise 5.8

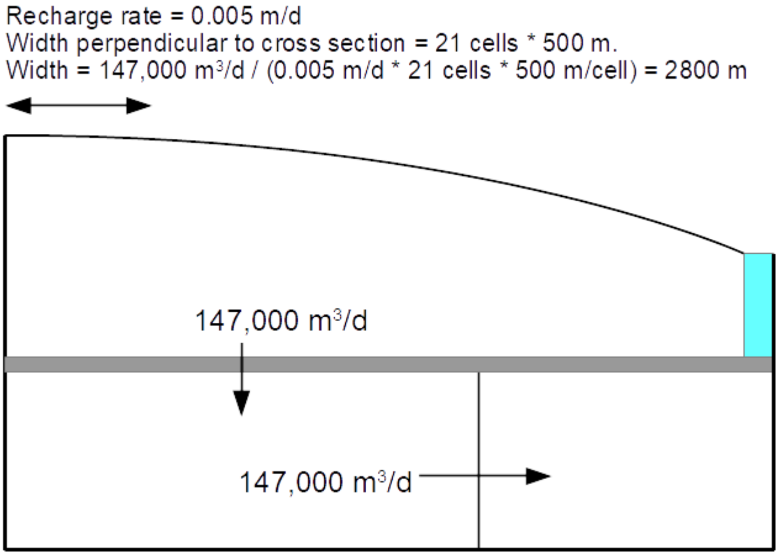
Solution to Exercise 5.9

From the solution to Exercise 5.8, the volume of flow that reaches layer 3 is known to be 147,000 m^3/d. The recharge rate is 0.005 m/d and the length of the system is 10,500 m. The required width of recharge is calculated here and illustrated visually in the image below.

$$Volumetric\ Flow = RechargeRate \times Length * Width$$

$$\frac{Volumetric\ Flow}{RechargeRate \times Length} = Width$$

$$\frac{147{,}000\ \frac{m^3}{d}}{0.005\ \frac{m}{d} \times 10{,}500\ m} = 2{,}800\ m$$



Diagrammatic cross section of model with area of recharge to the third layer highlighted with a double-headed arrow.

Return to Exercise 5.9

Solution to Exercise 5.10

In exercise 5.6, the river was represented by Constant Head package whereas in this exercise, it is represented by the River package. Recharge applied to a Constant Head cell is ignored but recharge on a River cell is included in the model, so Q_{In} includes a recharge term in this exercise. The additional flow into the groundwater is matched by additional flow out of the river in order to maintain mass balance. Notice that the head of cell (1,1,20) is 320.250 m in this exercise whereas the head in exercise 5.6 was 320.0. With a river boundary, the head in the cell must be higher than the river stage for flow into the river from the groundwater to occur. In this case, the river stage is equal to the top elevation of the cell so the head in the cell is higher than the top of the cell. The flow to the river can also be calculated by dividing the total flow to all the river cells (from the global water budget in the listing file) by the number of river cells. To calculate the internal flows Q_{West} and Q_{Bot}, the procedure in the solution to Exercise 3.1 is used.

Cell (1,1,20)

$$Q_{Out} = Q_{RIV}$$

$$Q_{In} = Q_{West} + Q_{Bot} + Q_{RCH}$$

$$Q_{RIV} = 100{,}000\frac{m^2}{d} \times (320.25\text{ m} - 320\text{ m}) = 25{,}000\ \frac{m^3}{d}$$

$$Q_{West} = 21{,}583\ \frac{m^3}{d}$$

$$Q_{Bot} = 2{,}189\ \frac{m^3}{d}$$

$$Q_{RCH} = 1{,}250\frac{m^3}{d}$$

$$Q_{In} = 21{,}583\frac{m^3}{d} + 2{,}189\ \frac{m^3}{d} + 1{,}250\ \frac{m^3}{d} = 25{,}022\frac{m^3}{d}$$

$$Q_{Out} = Q_{RIV} = 25{,}000\frac{m^3}{d} = \frac{flow\ to\ all\ river\ cells}{number\ of\ river\ cells} = \frac{525000\frac{m^3}{d}}{21}$$

$$Q_{Out} - Q_{In} = 25{,}000\frac{m^3}{d} - 25{,}022\frac{m^3}{d} = -22\ \frac{m^3}{d}$$

Balance Error = 0.10%

Return to Exercise 5.10

Solution to Exercise 5.11

The groundwater flows into cell (1,15,10) are calculated using Darcy's law. Recharge is computed by multiplying the cell area by the recharge rate. The drain flow is computed by multiplying the drain conductance by the difference between the drain elevation and the elevation of the head in the cell. The head in the cell can be determined by importing the simulated heads into ModelMuse and coloring or contouring the head data. The maximum drawdown caused by the drain occurs at its western end where the drawdown exceeds 19 m but every cell has at least a slightly lower head than in the model without the drain. To calculate the internal flows Q_{West}, Q_{Eest}, Q_{South}, Q_{North} and Q_{Bot}, the procedure in the solution to Exercise 3.1 is used.

Cell (1,15,10)

$$Q_{Out} = Q_{East} + Q_{DRN} \qquad Q_{In} = Q_{South} + Q_{West} + Q_{North} + Q_{Bot} + Q_{RCH}$$

$$Q_{DRN} = Conductance * \Delta h = 100{,}000\ \frac{m^2}{d} \times (323.186\text{ m} - 322.5\text{ m}) = 68{,}600\ \frac{m^3}{d}$$

$$Q_{East} = 1{,}705\ \frac{m^3}{d} \qquad Q_{South} = 19{,}469\frac{m^3}{d} \qquad Q_{West} = 27{,}683\ \frac{m^3}{d}$$

$$Q_{North} = 20{,}930\ \frac{m^3}{d} \qquad Q_{Bot} = 954\ \frac{m^3}{d} \qquad Q_{RCH} = 1{,}250\frac{m^3}{d}$$

$$Q_{Out} = 1{,}705\ \frac{m^3}{d} + 68{,}600\ \frac{m^3}{d} = 70{,}305\ \frac{m^3}{d}$$

$$Q_{In} = 19{,}469\frac{m^3}{d} + 27{,}683\frac{m^3}{d} + 20{,}930\frac{m^3}{d} + 954\frac{m^3}{d} + 1{,}250\frac{m^3}{d} = 70{,}286\frac{m^3}{d}$$

$$Q_{Out} - Q_{In} = 70{,}305\frac{m^3}{d} - 70{,}286\frac{m^3}{d} = 19\frac{m^3}{d}$$

$$\text{Balance Error} = 0.03\%$$

Cell (1,15,19)

Even though this cell contains a drain, the drain is inactive because the head in the cell is less than the drain elevation. The head in the cell is 321.7 which is less than the drain elevation of 322.5. When the cursor is held over the cell, the head will be displayed on the status bar. Alternatively, flows could be imported from the cell-by-cell budget file and the drain flows could be plotted to quickly identify inactive drains.

$$Q_{DRN} = 0$$

$$Q_{East} = 7{,}828\frac{m^3}{d} \qquad Q_{South} = 427\frac{m^3}{d} \qquad Q_{West} = 4{,}415\frac{m^3}{d}$$

$$Q_{North} = 687\frac{m^3}{d} \qquad Q_{Bot} = 1{,}042\frac{m^3}{d} \qquad Q_{RCH} = 1{,}250\frac{m^3}{d}$$

$$Q_{Out} = 7{,}828\,\frac{m^3}{d} + 0 = 7{,}828\,\frac{m^3}{d}$$

$$Q_{In} = 427\frac{m^3}{d} + 4{,}415\frac{m^3}{d} + 687\frac{m^3}{d} + 1{,}042\frac{m^3}{d} + 1{,}250\frac{m^3}{d} = 7{,}821\frac{m^3}{d}$$

$$Q_{Out} - Q_{In} = 7{,}828\frac{m^3}{d} - 7{,}821\frac{m^3}{d} = 7\frac{m^3}{d}$$

$$\text{Balance Error} = 0.09\%$$

Return to Exercise 5.11↥

Solution to Exercise 5.12

In run PS2C the drains in columns 19 and 20 are inactive because the heads in those cells drop below the elevation of the drain (322.5). When the drain is replaced by a general head boundary, the GHB features in columns 19 and 20 remain active and act as a source of water to those cells because the boundary head of the GHB is larger than the head in the cell, which indicates the GHB acts as a source of water for those cells. By comparing the water budgets for PS2C and PS2C1, it can be seen that the GHB boundary has flow both into and out of the groundwater system whereas the drain in PS2C has only outflowing water.

Return to Exercise 5.12↥

Solution to Exercise 5.13

The total volumetric rate of flow to the river in run PS2D decreases by 26,353 m^3/d compared with that of run PS2C. The total volumetric rate of flow to the drain in run PS2D decreases by 48,647 m^3/d compared with that in run PS2C. The combined change in discharge to the river and the drain equals the well discharge rate.

Flow rate out	PS2D	PS2C	PS2D - PS2C
River	269,979	296,332	-26,353
Drain	180,021	228,668	-48,647
Well	75,000	0	75,000
			Sum = 0

By comparing the head in run PS2D and PS2C, the drawdown in the vicinity of the well is 8.35 m. It is important to note that this is the average drawdown for the cell containing the well and drawdown in the well itself is larger.

Return to Exercise 5.13↥

Solution to Exercise 5.14

The heads are lower everywhere but especially in the vicinity of the two wells. The flows out of the groundwater system to both the river and drain are reduced. The total reduction, compared to the flows in model PS2D, is equal to the flow out of the well that was added in layer 3. The additional drawdown at the first well (in layer 1) is 2.2 m. The drawdown at the second well (in layer 3) is 5.4 m. These drawdowns are computed by comparing the heads in models PS2D and PS2D1.

Save your sketch for later comparison with the results that will be generated in the MODPATH problem set.

Return to Exercise 5.14↥

Solution Exercise 6

Solution to Exercise 6.1

Model PS3A	**Flow rate (round to the nearest whole number)**		
Budget component	Steady-state, no well (period 1, step 1)	Steady-state with well (period 3, step 1)	Change in flow rate (with well - without well)
	Inflow		
Storage-specific storage	0	0	0
Storage-specific yield	0	0	0
Constant head	0	72,756	72,756
Wells	0	0	0
River leakage	0	0	0
Recharge	237,500	237,500	0
Total	237,500	310,256	72,756
	Outflow		
Storage-specific storage	0	0	0
Storage-specific yield	0	0	0
Constant head	0	0	0
Wells	0	100,000	100,000
River leakage	237,500	210,256	-27,244
Recharge	0	0	0
Total	237,500	310,256	72,756
Inflow - outflow	0		0

Return to Exercise 6.1↥

Solution to Exercise 6.2

Model PS3A	**Flow rate**					
Budget component	Period 1, step 1	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow						
Storage-specific storage	0	38	37	30	10	2
Storage-specific yield	0	99,921	97,307	79,248	25,829	5,219
Constant head	0	32	1984	14,938	53391	68,781
Wells	0	0	0	0	0	0
River leakage	0	0	0	0	0	0
Recharge	237,500	237,500	237,500	237,500	237,500	237,500
Total inflow	237,500	337,491	336,827	331,716	316730	311,503
Outflow						
Storage-specific storage	0	0	0	0	0	0
Storage-specific yield	0	0	0	0	0	0
Constant head	0	0	0	0	0	0
Wells	0	100,000	100,000	100,000	100,000	100,000
River leakage	237,500	237,491	236,827	231,716	216,730	211,503
Recharge	0	0	0	0	0	0
Total outflow	237,500	337,491	336,827	331,716	316,730	311,503
Inflow - outflow	0	0	0	0	0	0

Model PS3A	**Change in flow rate**[1]				
Budget component	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow					
Storage-specific storage	38	37	30	10	2
Storage-specific yield	99,921	97,307	79,248	25,829	52,129
Constant head	32	1984	14,938	53,391	68,781
Wells	0	0	0	0	0
River leakage	0	0	0	0	0
Recharge	0	0	0	0	0
Total inflow change	99,991	99,327	94,216	79,230	74,003
Outflow					
Storage-specific storage	0	0	0	0	0
Storage-specific yield	0	0	0	0	0
Constant head	-33	-521	-922	-1133	-1136
Wells	100,000	100,000	100,000	100,000	100,000
River leakage	-9	-673	-5784	-20,770	-25,997
Recharge	0	0	0	0	0
Total outflow change	99,991	99,327	94,216	79,230	74,003
Inflow - outflow	0	0	0	0	0

[1] Rate at the current time step minus the initial rate at period 1, step 1. Round to the nearest whole number.

Return to Exercise 6.2↥

Solution to Exercise 6.3

Model PS3B	**Flow rate (round to the nearest whole number)**		
Budget component	Steady-state, no well (period 1, step 1)	Steady-state with well (period 3, step 1)	Change in flow rate (with well - without well)
Inflow			
Storage-specific storage	0	0	0
Storage-specific yield	0	0	0
Constant head	0	0	0
Wells	0	0	0
River leakage	0	0	0
Recharge	237,500	237,500	0
Total	237,500	237,500	0
Outflow			
Storage-specific storage	0	0	0
Storage-specific yield	0	0	0
Constant head	0	0	0
Wells	0	100,000	100,000
River leakage	237,500	137,500	-100,000
Recharge	0	0	0
Total	237,500	237,500	0
Inflow - outflow	0	0	0

Without the specified heads in Exercise 6.1, the total inflow is fixed so the difference is made up by a greater decrease in flow to the river.

Return to Exercise 6.3⮥

Solution to Exercise 6.4

Model PS3B	**Flow rate**					
Budget component	Period 1, step 1	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow						
Storage-specific storage	0	38	38	35	25	15
Storage-specific yield	0	99,953	99,263	93,358	66,980	42,034
Wells	0	0	0	0	0	0
River leakage	0	0	0	0	0	0
Recharge	237,500	237,500	237,500	237,500	237,500	237,500
Total inflow	237,500	337,491	336,801	330,894	304,505	279,551
Outflow						
Storage-specific storage	0	0	0	0	0	0
Storage-specific yield	0	0	0	0	0	0
Wells	0	100,000	100,000	100,000	100,000	100,000
River leakage	237,500	237,491	236801	230,894	204,505	179,551
Recharge	0	0	0	0	0	0
Total outflow	237,500	337,491	336,801	330,894	304,505	279,551
Inflow - outflow	0	0	0	0	0	0

Model PS3B	**Change in flow rate**[1]				
Budget component	Period 2, step 1	Period 2, step 3	Period 2, step 5	Period 2, step 8	Period 2, step 10
Inflow					
Storage-specific storage	38	38	35	25	15
Storage-specific yield	99953	99263	93358	66980	42034
Wells	0	0	0	0	0
River leakage	0	0	0	0	0
Recharge	0	0	0	0	0
Total inflow change	99991	99301	93394	67005	42051
Outflow					
Storage-specific storage	0	0	0	0	0
Storage-specific yield	0	0	0	0	0
Wells	100,000	100,000	100,000	100,000	100,000
River leakage	-9	-699	-6606	-32995	-57949
Recharge	0	0	0	0	0
Total outflow change	99991	99301	93394	67005	42051
Inflow - outflow	0	0	0	0	0

[1] Rate at the current time step minus the initial rate at period 1, step 1. Round to the nearest whole number.

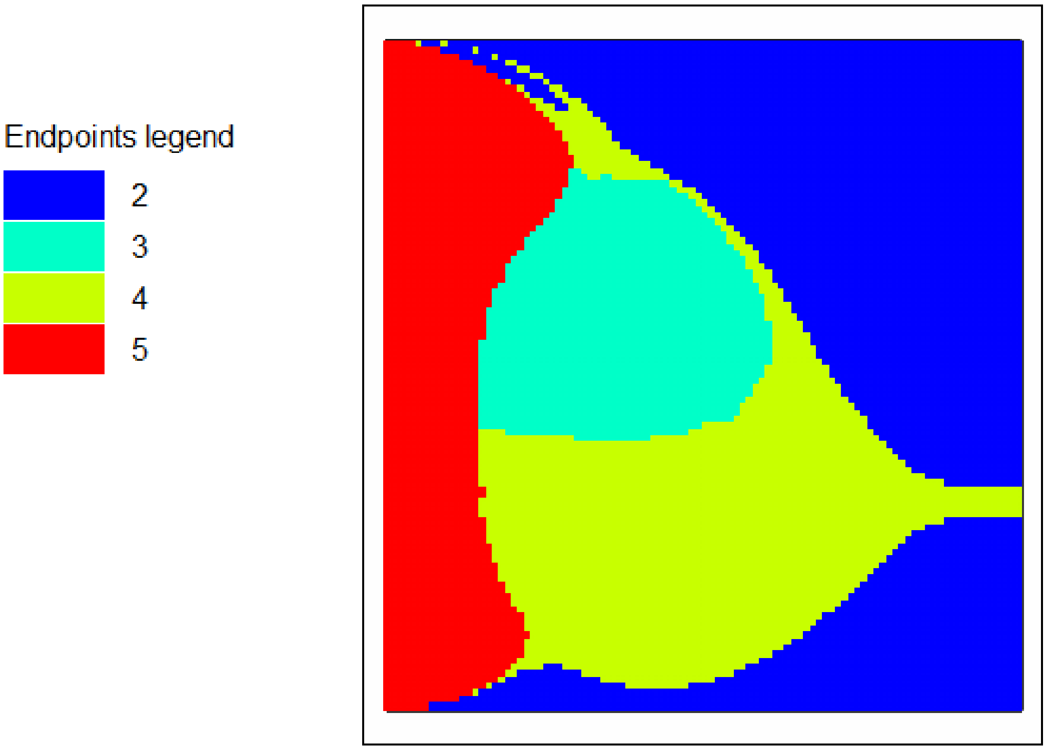
In PS3B, the system has system is closer to reaching equilibrium. The absolute change in river outflow is greater while the release from storage is less.

Return to Exercise 6.4↑

Solution Exercise 7

Solution to Exercise 7.1

The results should look similar to the image below.

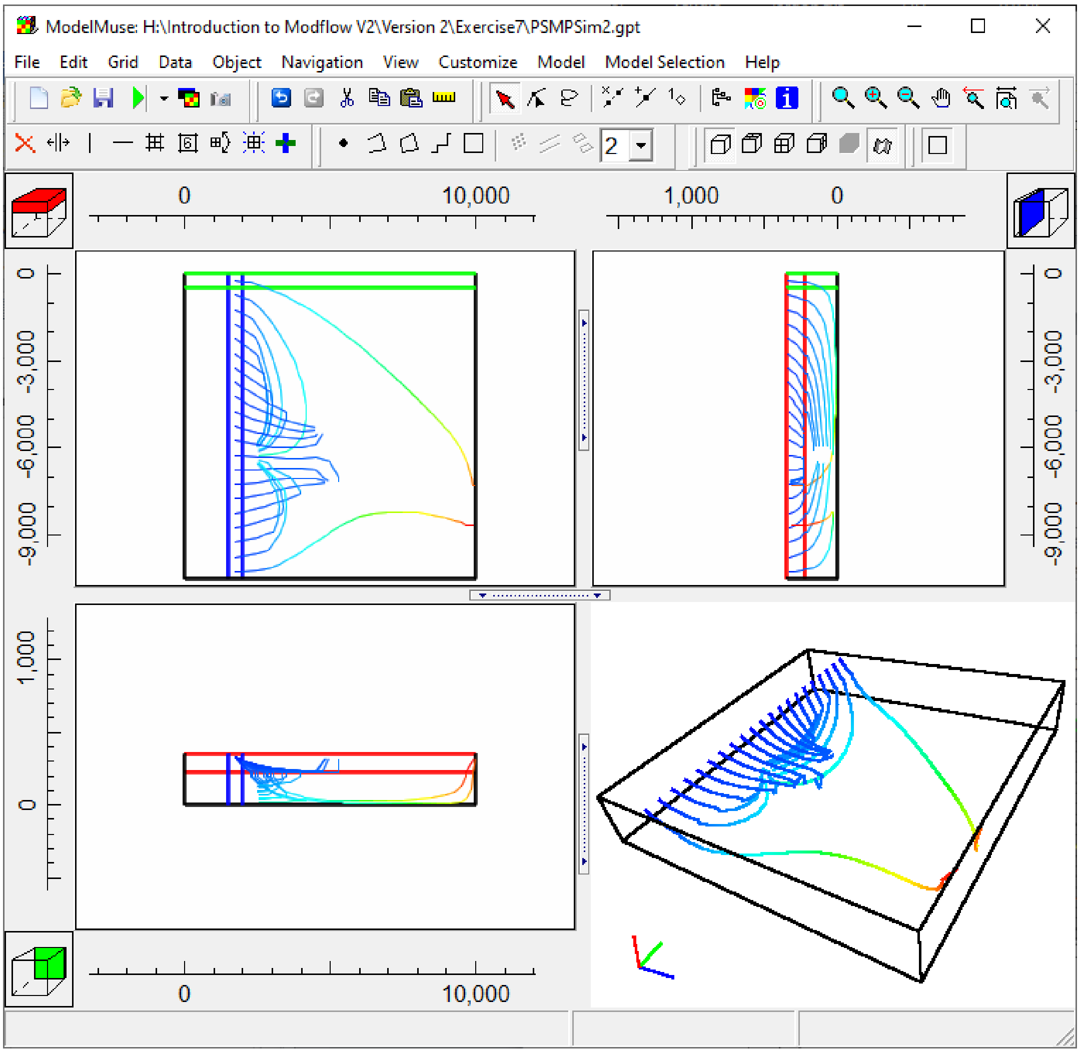


Plot of MODPATH ending zones plotted at particle starting locations.

Return to Exercise 7.1

Solution to Exercise 7.2

The results will look similar to the image below.

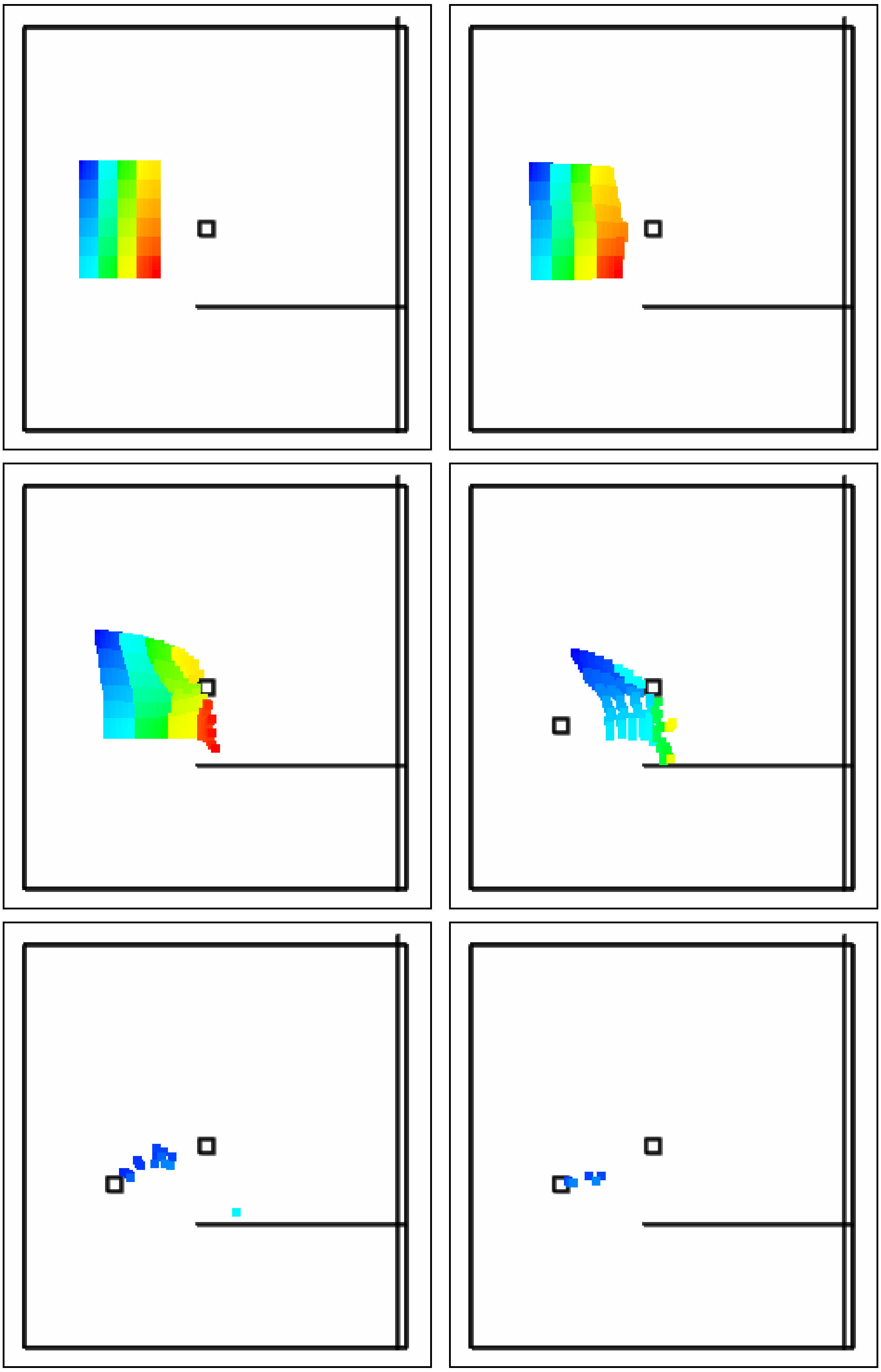


Plot of pathlines.

[Return to Exercise 7.2](#)

Solution to Exercise 7.3

Plots for the particles at selected times are shown in the image below. Once a particle exits the model through a boundary condition it is no longer tracked by MODPATH.



Plots of MODPATH particles at times 0, 1000, 4000, 9000, 19,000, and 24,000.

Return to Exercise 7.3↥

Solution Exercise 8

Solution Exercise 8.1

A solution is not provided for Phase 1 of Exercise 8 because models are a simplification of reality and there is no assessment that can determine if a model is the correct representation of the system. The site information includes a statement that provides a clue for improving the model constructed by following the steps of Exercise 8. It says: "The gravels closest to the current river channel are coarsest and contain less fine material than the rest of the alluvium." Thus, the model might be improved by increasing the hydraulic conductivity close to the stream or reducing it further away from the stream. An alternative example model named AlluviumSSHOB_ModifiedK.mmZLib is provided with the files for this exercise. In this version of the model, the hydraulic conductivities at TW-3 and TW-8 have been reduced to 10. This results in a somewhat improved match in the simulated and observed heads.

Return to Exercise 8.1↥

Solution Exercise 8.2

If the production well is at the location of TW-4, the minimum travel time from the river is greater than 60 days so TW-4 is acceptable. However, the minimum travel time is less than 365 days. Some other potential well locations, such as at row 46 column 70, provide a minimum travel time greater than 365 days.

Click to return to Exercise 8.2↥

18 About the Author

Richard Winston is a hydrologist in the Integrated Modeling and Prediction Division of the United States Geological Survey. He is an author of several computer programs to aid in the development and understanding of groundwater models. His staff profile can be viewed at this link↗.

Please consider signing up for the Groundwater Project mailing list to stay informed about new book releases, events, and ways to participate in the Groundwater Project. When you sign up for our email list, it helps us build a global groundwater community. Sign-up↗.

THE GROUNDWATER PROJECT

Modifications to Original Release

Changes from the Original Version to Version 2

Original Version: November 7, 2023, Version 2: November 8, 2023

Page numbers refer to the original PDF.

page 27, corrected the order of Figure 12 part a) and b) – what was b) is now a) and vice versa.

Changes from Version 2 to Version 3

Version 2: November 8, 2023, Version 3: November 20, 2023

Page numbers refer to the Version 2 PDF.

page i, added message regarding support of The Groundwater Project

page ii, changed version number from 2 to 3

page 94, Exercise 4.1, the link to Box 7 was corrected

page 94, Exercise 4.2, the link to Box 8 was corrected

page 94, Exercise 4.3, the link to Box 9 was corrected

page 95, Exercise 4.4, the link to Box 10 was corrected

page 96, Exercise 5 Part A, the link to Box 11 was corrected

Changes from Version 3 to Version 4

Version 3: November 20, 2023, Version 4: December 31, 2023

Page numbers refer to the Version 3 PDF.

page ii, updated version number and date page iii, added page requesting support of the Groundwater Project

page iii, now page iv. corrected year from 2022 to 2023 in the APA citation

page iii, now page iv, added "Any use of trade, firm, or product names is for descriptive purposes only and does not imply endorsement by the U.S. Government."

page 13, corrected page numbering of main text to start at 1, so all subsequent page numbers decreased by 12

Changes from Version 4 to Version 5

Version 4: December 31, 2023, Version 5: January 30, 2024

Page numbers refer to the Version 4 PDF.

page iii, updated version number and date

page xiii, added Sean Boemer to acknowledgments

page 86, corrected link to Box 12 for instructions for item 4 of Part B of Exercise 5

page 88, corrected link to Box 13 for instructions for item 5 of Part C of Exercise 5

page 151, changed HClose to DVClose in last sentence of item 8 Box 5

page 160, changed 4 to 3 in last line of item 5 in Box 6

page 161, caption Figure Box 6-5, changed "zone 2 to zone 4" to "zone 3 to zone 4"

page 173, corrected the link for returning to where the text links to Box 12

page 226, Solution to Exercise 5.6, second paragraph, second sentence, Q_{East} was corrected to Q_{West}

page 227, corrected value in calculation of Q_{Bot} of Solution to Exercise 5.6 by changing the head in the cell of layer 2 from 324.227 to 328.769 with the result changing from 2181.343 to 2181.289

Changes from Version 5 to Version 6

Version 5: January 30, 2024, Version 6 February 23, 2024

Page numbers refer to the Version 5 PDF.

page iii, updated version number and date

page 200, revised Figure Box 20-5 to match the description of the figure

page 213, added a sentence explaining the disappearance of some MODPATH particles

page 239, added a sentence explaining the disappearance of some MODPATH particles

page 239, slightly reduced size of figure