# Table of Contents

## Part 1 Introduction
1.1 Installation and Licensing ................................................................. 3
1.2 Features .......................................................................................... 4
1.3 What’s New
   1.3.1 Version 6.0 ............................................................................. 10
   1.3.2 Version 5.1 ............................................................................. 17
   1.3.3 Version 5.0 ............................................................................. 20
   1.3.4 Version 4.1 ............................................................................. 26
1.4 How to buy ...................................................................................... 30

## Part 2 Program Overview
2.1 Main Menus and Toolbars ................................................................. 36
   2.1.1 Preferences ........................................................................... 39
   2.1.2 Project Settings .................................................................... 43
2.2 Project Version Compatibility .......................................................... 49
2.3 Keyboard Shortcuts .......................................................................... 51

## Part 3 Quick Start Tutorials
3.1 Conceptual Modeling Tutorial ............................................................ 55
3.2 Importing VMOD/MODFLOW Models ............................................... 98
3.3 Airport Numerical Model with Transport ......................................... 120
3.4 SEAWAT Tutorial ........................................................................... 186
3.5 PEST with Pilot Points .................................................................... 219
3.6 MODFLOW-USG Tutorial ............................................................... 243

## Part 4 Working with Your Data
4.1 Importing Data ................................................................................ 280
   4.1.1 Points ..................................................................................... 280
   4.1.2 Polylines ................................................................................. 290
   4.1.3 Polygons ................................................................................. 295
   4.1.4 Surfaces ................................................................................ 296
       SURFER Grid Files ..................................................................... 298
       ESRI ASCII Raster Files ............................................................ 301
   4.1.5 Wells ...................................................................................... 304
   4.1.6 3D Gridded Data ................................................................... 312
       TecPlot 3D Point Grid File .......................................................... 316
   4.1.7 HGA Cross-Sections .............................................................. 318
   4.1.8 Maps ...................................................................................... 320
   4.1.9 Time Schedules .................................................................... 328
   4.1.10 DXFs ................................................................................... 331
4.2 Importing VMOD/MODFLOW Models ............................................. 331

© 2019 by Waterloo Hydrogeologic
Part 9 Converting Conceptual Models to Numerical Models

9.1 Converting to MODFLOW-2000/2005 ................................................................. 526
9.2 Converting to MODFLOW-USG ................................................................. 530
9.3 Converting to FEFLOW .............................................................................. 533

Part 10 Numerical Modeling Workflow - Finite Difference Grids

10.1 Define Objectives ......................................................................................... 539
  10.1.1 Project Property Settings (Flow) .............................................................. 548
  10.1.2 Species Parameters ........................................................................... 550
  10.1.3 Model Parameters ........................................................................... 556
10.2 Define Numerical Model ............................................................................ 561
10.3 View/Edit Grid ............................................................................................. 566
10.4 Define Properties ........................................................................................ 570
  10.4.1 Theory ............................................................................................... 577
10.5 Define Boundary Conditions ...................................................................... 597
  10.5.1 Edit Attributes in TXT Format .............................................................. 612
  10.5.2 Transport ............................................................................................ 613
  10.5.3 Wells .................................................................................................... 615
  10.5.4 Theory ............................................................................................... 627
10.6 Define Observations .................................................................................... 628
10.7 Define Zone Budget Zones ......................................................................... 632
10.8 Define Particles .......................................................................................... 636
10.9 Select Engines ............................................................................................ 644
10.10 Translation Settings .................................................................................. 645

  10.10.1 MODFLOW ....................................................................................... 646
        General ................................................................................................. 647
        MODFLOW Settings ........................................................................... 647
        LGR settings ....................................................................................... 650
        Solvers ................................................................................................. 651
        PCG ...................................................................................................... 651
        GMG ................................................................................................... 653
        WHS ................................................................................................... 655
        SIP ...................................................................................................... 656
        SOR ...................................................................................................... 657
        SAMG ................................................................................................ 658
        NWT ................................................................................................... 660
        Recharge and Evapotranspiration .......................................................... 662
        Lakes .................................................................................................. 663
        Layer Types ........................................................................................ 665
        Cell Rewetting ..................................................................................... 666
        Initial Heads ....................................................................................... 669
Part 11 Numerical Modeling Workflow - Unstructured Grids (MODFLOW-USG) 745
11.1 Define Properties ................................................................. 746
11.2 Define Boundary Conditions .................................................. 748
11.3 Define Observations ............................................................... 752
11.4 Define Zone Budget Zones - USG ........................................... 757
11.5 Define Particles ................................................................. 761
11.6 Select Engines ................................................................. 761
11.7 Translation Settings ............................................................. 763
  11.7.1 MODFLOW-USG ............................................................ 764
    Solvers ............................................................................. 768
    Initial Heads ..................................................................... 774
    Output Control .................................................................. 776
    Advanced Settings .......................................................... 778
  11.7.2 MOD-PATH3DU Translation Settings ............................... 781
11.8 Run Engines ................................................................. 784
11.9 View Results ................................................................. 784
  11.9.1 View Maps (USG) .......................................................... 785

Part 12 Expression Builder ......................................................... 787

Part 13 PEST (Parameter Estimation) Workflow ..................... 794
  13.1 Define Observations .......................................................... 796
  13.2 Define Parameters ............................................................. 798
13.3 Define Pilot Points ................................................................. 801
13.4 Define Kriging Variograms .................................................... 804
13.5 Select Run Type ........................................................................ 807
   13.5.1 Run Sensitivity Analysis .................................................. 808
   13.5.2 Select Regularization ...................................................... 810
      No Regularization............................................................... 812
      Tikhonov Regularization.................................................... 813
      SVD Assist........................................................................... 814
13.6 Run PEST ............................................................................. 817
13.7 Analyze Results ................................................................. 819
13.8 Save PEST Parameters as New Inputs ....................................... 820
13.9 Making Changes after a PEST Run ........................................... 821

Part 14 Working with Multiple Model Scenarios 823
Part 15 VMOD Flex and VMOD Classic 827
Part 16 References 844
Part 17 Appendix A - Input files and Packages 851
Part 18 Appendix B - VM Flex GUI and Input Files 855
Index 860
1 Introduction

Visual MODFLOW (VMOD) Flex is a powerful software package that provides the tools for building three-dimensional groundwater conceptual and numerical models using raw GIS data objects. The conceptual model approach to groundwater modeling allows you to:

- **Build a conceptual model of the groundwater system, prior to the simulation** - Geologic formation structures, hydrogeologic properties, and boundary conditions are all designed outside the model grid or mesh; this allows the flexibility to adjust your interpretation of the groundwater system before applying a discretization method and converting to a numerical model.

- **Build the model with minimal data pre-processing required** - Working with grid-independent data allows you to maximize the use of your existing GIS data and incorporate physical geology and geographic conditions before designing a grid or mesh.

- **Generate and simulate regional and local-scaled models** - With support for MODFLOW-LGR, you can design local grids around areas of interest, directly within the conceptual model environment. Calculated heads from a regional model can also be used as boundary conditions for local-scaled models.

- **Design the correct model faster** - The grid-independent raw data is left intact and is not constricted by grid cells or mesh elements when modifying the data and project objective. This allows you to generate multiple numerical models from the same conceptual model.

- **Make changes to the model data and immediately see results** - The conceptual model environment provides simultaneous 2D and 3D views which are updated whenever changes to the data are made.

This document provides detailed descriptions of all features and functionality available in Visual MODFLOW Flex.

💡 How to get started

- Study this Introduction chapter and Program Overview sections to familiarize yourself with the program.

- Then work through the Quick Start Tutorials to familiarize yourself with using VMOD Flex. These sections cover importing MODFLOW and VMOD Classic projects as well as building...
new models using the Conceptual Modeling approach.

Learning more

➤ See Conceptual Modeling Workflow for instructions on building a grid-independent conceptual model using your raw data.

➤ See Numerical Modeling Workflow for more instructions on working with numerical inputs, translating to MODFLOW packages, and running and analyzing the results.

➤ See Working with Multiple Model Scenarios for a better understanding of the various workflows in VMOD Flex.

➤ See the Program Overview section for detailed instructions on using VMOD Flex.

➤ Looking for help beyond what's in this manual? We have many additional resources for you:

✓ LinkedIn User’s Group
✓ Visual MODFLOW Flex FAQ
✓ An Introduction to Groundwater Modeling Concepts
✓ Getting Started with Visual MODFLOW Flex
✓ Frequently Asked Questions
✓ Your Complete Guide to MODFLOW-USG
✓ All Tutorials
✓ Visual MODFLOW Flex ReadMe
✓ White Papers & Technical Articles
1.1 Installation and Licensing

Hardware Requirements

To run the latest version of Visual MODFLOW Flex, you will need the following minimum system configuration:

<table>
<thead>
<tr>
<th>Operating System</th>
<th>Windows 10 (Pro, Enterprise)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Windows 8 (Professional, Enterprise, Ultimate)</td>
</tr>
<tr>
<td></td>
<td>Windows 7 (Professional, Enterprise, Ultimate)</td>
</tr>
</tbody>
</table>

**NOTE:** Administrative rights may be required to install the software

<table>
<thead>
<tr>
<th>Processor</th>
<th>32-bit or 64-bit (Pentium 4 or higher)</th>
</tr>
</thead>
<tbody>
<tr>
<td>RAM</td>
<td>8GB or more recommended</td>
</tr>
<tr>
<td>Hard Disk</td>
<td>1.5 GB Free Space, plus extra space for your models</td>
</tr>
<tr>
<td>Networking Hardware</td>
<td>Network Card (required for soft key licensing)</td>
</tr>
<tr>
<td>Software</td>
<td>MS-Office (2000 or later). If you do not have Office, you must install the Access Database Engine</td>
</tr>
</tbody>
</table>

**Please Note:** If you intend to build complex projects, it is recommended that you upgrade beyond the specifications in the above list.

Software Requirements

To run the latest version of Visual MODFLOW Flex, the following software pre-requisites are required:

- Microsoft .NET Framework v.4.0 installed (provided with installation)
- Microsoft Office 2000 (or later) or Microsoft Access Database Engine

**NOTE:** Microsoft Office 64-bit version is currently not supported for XLS and MDB import.
If you have any problems with your particular system configuration, please contact your system administrator, or technical support: support@waterloohydrogeologic.com

Installation and Licensing

When you purchase Visual MODFLOW Flex, you will receive download and installation instructions. Please consult these for the latest instructions.

Starting Visual MODFLOW Flex

Once Visual MODFLOW Flex has been installed on your computer, simply double-click on the Visual MODFLOW Flex shortcut icon located on your computer’s desktop.

Alternatively, you can access the software via the start menu by clicking on Start/Programs/...

Please Note: If you are using dongle-based hardware licensing, please ensure that your dongle is connected to your computer AFTER you have installed the software, and that you have properly configured your installation.

1.2 Features

Visual MODFLOW Flex is a graphical user interface for developing, running, and visualizing groundwater simulations that brings together industry-standard codes for groundwater flow and contaminant transport, essential analysis and calibration tools, and stunning 3D visualization capabilities in a single, easy-to-use software environment.

Work With Geographic Data

Visual MODFLOW Flex supports the following coordinate systems:
- Geographic coordinate systems (data import only)
- Projected coordinate systems: UTM, StatePlane
- Local Cartesian

Work With Grid-Independent Data

Import spatial and attribute data from a wide variety of data types including:
- Points (.XLS, .TXT, .CSV, .MDB, .SHP, .DXF, .TRP)
- Polylines (.SHP, .DXF)
- Polygons (.SHP, .DXF)
- 3D Gridded Data (.HDS, .DAT)
- Raster Images (.BMP, .TIF, .JPG)
• Time Schedules (.XLS)
• Surfaces (.DEM, .GRD, .TXT, .ASC)
• Hydro GeoAnalyst (HGA) Cross Sections (.3XS)
• Vertical and Horizontal Wells (.XLS, .CSV)

View and modify settings for imported data:
• View data object meta data including the source file name, field mappings and the native coordinate system
• View raw attribute data in a spreadsheet view
• Apply mathematical operations to data, e.g., set an attribute as a constant value, convert well tops to a points data object, and convert HGA cross section model layers to a points data object
• Drape a raster image over a surface data object, e.g., digital elevation models
• Set symbol properties for points, polygons, polylines and display labels using a variety of style options
• Color render shape features by attribute value using a classified or stretched color scheme
• Show contour lines and set color rendering options for surface layers
• Add, remove and modify wells and associated well data including screens intervals, diver observation points, well tops, well paths (for horizontal only), and pumping schedules

Create surfaces from points data objects:
• Using one or more point data objects, generate surface layers using Inverse Distance, Kriging or Natural Neighbor interpolation methods
• Configure the interpolation method by modifying various interpolation settings
• Clip the generated surface to the horizontal extents of a specified polygon data object

Digitize new data objects using 2D Viewer:
• Using the 2D Viewer editing tools, digitize a new polyline, polygon or points data object

2D & 3D Visualization

Visualize data objects and conceptual model features using interactive 2D and 3D Viewers:
• Use various screen configurations to display multiple 3D or 2D Viewers simultaneously
• Zoom, rotate and move data within the viewer using your mouse or keyboard
• Modify viewer settings including the background color and vertical exaggeration (3D Viewer only)
• In 3D Viewers, remove parts of the displayed data by creating cutaways along the X, Y and Z axis
• In 2D Viewers, select individual data object features (points, line, shapes), and then view the corresponding attribute data in spreadsheet view, and vice versa

Edit data object geometry in 2D Viewer:
• Modify existing data objects by manually digitizing points, polylines and polygons
• Rotate, scale and delete shapes
• "Undo" all edits and revert back to original shape

---

**Define Multiple Conceptual Models**

Create multiple conceptual models:
• Create multiple conceptual models with different interpretations, or copy existing conceptual models
• Define conceptual model geometry using imported data objects
• Define the horizontal model boundary using an imported or digitized polygon data object
• Create vertical horizons from surfaces that are either imported or created by interpolating raw XYZ points
• Select from different horizon types to accommodate various geological conditions (pinchouts, discontinuous layers, etc.)
• Automatically create 3D structural zones from defined horizons

---

**Property Modeling**

• Create property zones from imported or digitized polygon data objects, or from generated structural zones
• Assign property values for conductivity, storage and initial heads using various methods:
  o Use a constant value
  o Map to imported polygon shapefile attributes
  o Map to imported 3D Gridded data attributes
  o Use surface data object

---

**Boundary Condition Modeling**

Create Boundary Conditions

• Automatically generate the simulation domain using the boundaries defined for the conceptual model
• Apply boundary conditions to the top, bottom, sides or an intermediate layer of the simulation model domain
• Support for the following groundwater flow and transport boundary conditions:
  o Drain (DRN)
  o Evapotranspiration (EVT)
  o Flow and Head Boundary/Specified Flux (FHB)
  o General Head (GHB)
  o Horizontal Flow Barrier/Wall (HFB)
  o Lake (LAK)
  o Pumping Wells (WEL)
  o Recharge (RCH)
Recharge Seepage Face (RSF) (*MODFLOW-SURFACT only*)
- River (RIV)
- Specified Concentration (SSM)
- Specified Head (CHD)
- Time Varying Material Properties (TMP) (*MODFLOW-SURFACT only*)
- Unsaturated Zone (UZF)

- For linear boundary conditions, define local zones from line segments using an interactive 2D Viewer window
- For linear boundary conditions, define parameters at starting, ending, or intermediate vertices along a line and interpolate values between each vertex
- Set each boundary condition parameter as static or transient
- Define boundary condition parameters using one or more of the following methods:
  - Use a constant value
  - Map to imported shapefile attributes
  - Use a surface data object
  - Use a time schedule data object (for transient boundary conditions)
  - Use attributes from 3D Gridded data objects

## Model Discretization

Discretize your model using finite-difference, finite-element, or control volume methods:

- When working with finite-difference grids:
  - Specify the number of rows and columns, grid origin, and the angle of rotation
  - Choose from the following finite difference grid types: Deformed, Uniform, Semi-Uniform
  - Perform horizontal grid refinement/coarsening within a user-defined row/column interval
  - Define a child grid within a numerical grid for running Local Grid Refinement (LGR) simulations using the MODFLOW-LGR flow engine

- When working with finite-elements meshes:
  - Use imported shape data objects to define the super-element mesh
  - Choose from various Delaunay triangulation methods including constrained and conforming
  - Refine areas of the mesh using digitized or imported polygon shapes
  - Fit the mesh to your model domain using deformed or semi-uniform vertical slices

- When working with unstructured grids:
  - Define unstructured Voronoi and/or Quad-tree grids based on imported and created grid-independent feature objects (see related section above)
  - Refine areas of the grid around wells and boundary conditions
  - Accommodate pinchout layers when generating the numerical model, for accurate representation of the true site characteristics
1.3 What's New

Version 6.1 - June 2019

Usability and Other Enhancements

- Boundary Condition in/outflows from the Budget file are available in 2D/3D views and the cell inspector via the Output / Budget node in the model Explorer.

- You can Assign/Edit Mode Properties and Boundary Conditions using an input surface, horizon, or water table object in Expression Builder.
• Expression Builder includes round and significant digit functions

• You can zoom to an object in the Model Explorer in a 2D viewer using the context menu
• The SAMG solver is available with MODFLOW-USG
• You can specify ground surface or heads from a previous model run instead of using the initial head property values at the translation step for MODFLOW-USG runs
• Visual MODFLOW Flex warns you and allows you to back up a project before irreversibly upgrading projects created in previous versions
Defects Addressed

- Adding a data object such as a bitmap that is non-transparent during the grid creation process will overlay it on top of the grid rendering it invisible.
- Row/Column views for rotated grids are projected on NS/EW planes rather than on rotated model co-ordinate axes.
- In some cases, surfaces generated using the model domain polygon did not fully span the model domain due to differences in numerical precision.
- LST file takes a long time to print for transient models.
- Unhandled Exception when viewing certain .DXF files with unsupported components in a 2D Viewer.
- Performance Issues on Project Load and Reload.
- In some cases, not all observations are included when exporting data from the Calc. vs Obs. Chart.
- In some cases, linear boundary condition features were skipped during conceptual to numerical model conversion.
- Concentration output nodes are not always created in RT3D runs.
- Conceptual BCs assigned to the model sides not assigned as expected following conceptual to numerical model conversion to a finite difference grid if the grid cells are thin and/or there are steep gradients at the model edge.
- Uncaught exception visualizing Fluxes when budget file is removed.
- View Maps not showing heads output.
- Can't assign wells using Wells data object on Q-Grid.
- Well Edit form dives under the main Flex window.
- Only the first stress period values for bed leakance were included in LAK package translation.

For full version history, see the Visual MODFLOW Flex readme file at: https://www.waterloohydrogeologic.com/visual-modflow-flex-readme/

1.3.1 Version 6.0

Working with Unstructured Grid Models

- Build your unstructured MODFLOW-USG model using the same workflow steps and tools as in a structured Finite Difference model:
  - Define Modeling Objectives
  - Define Model Properties
  - Add Boundary Conditions
  - Define ZoneBudget Zones
  - Define Particles
• Define Quadtree Grids (Q-Grids)

• Refine vertical layers when creating unstructured grids
- Make use of advanced translation settings which provide greater flexibility in how your model simulations are formulated
- Support for the Ghost Node Correction (GNC) package
- Flex is now installed with MODFLOW-USG v1.4, the latest official USGS version as of the release date of Flex 6.0.

### Particle Tracking

- Support for mod-PATH3DU, which enables particle tracking in unstructured grid (MODFLOW-USG) models:

  ![Particle Tracking Image](image)

- Enhanced particle tracking capabilities for MODPATH and mod-PATH3DU allowing you to define:
  - Groups of particles that can be tracked and visualized separately
  - Separate particle release times for each particle group
  - Locations of particles based on existing or drawn point, polyline, polygon, or well features
- Particles in the cross-section (row/column) views of Finite Difference grids
- Particle elevations based on layer(s), surface(s), or a specified constant elevation

*Note:* mod-PATH3DU is a free utility developed by S.S. Papadopulos, Inc. (SSPA) that must be downloaded and installed separately from the SSPA website. Version 2.1 and higher are supported. Note, that Version 3.0 includes improvements to handle a wider variety of unstructured grids

---

### ZoneBudget

- ZoneBudget is now supported in unstructured MODFLOW-USG models
You can use point, polyline, or polygon features to define Zone Budget zones and Zone IDs can be specified using feature attributes.

**Visualization and Settings**

- Visualize velocity vectors in finite difference and unstructured grids. View:
  - In-plane velocity vectors
  - Color maps of out-of-plane velocities
Visual MODFLOW Flex 6.1

- Display average or Darcy velocities (total or in-plane) or by direction

- Specify a line feature along which to show cross-sectional color maps of unstructured grid model properties and outputs

- Cell Inspector now includes Layer/Model Top and Bottom elevations, and the magnitude and X-, Y-, and Z-components of the average/Darcy groundwater velocity

- When creating horizons, you can preview how the rules will be enforced in the 3D Viewer rather than viewing the raw surfaces.

### Packages, Properties, and Boundary Conditions

- Field calculator/expression builder for editing model properties and boundary condition includes:
  - cell geometry variables: Cell Bottom ($BOT), Model Bottom ($MBOT), Cell Thickness ($DZ), Cell Width ($DX), and Cell Length($DY) and
  - additional math functions: LOG, LOG10, NOT, ABS, DOT

- Copy Model (Flow/Transport) Properties and Boundary Conditions to from one Layer, Row, or Column to other Layers, Rows, or Columns

- Edit cells (active/inactive, zone budget) in row/column views.

### Data Import/Export

- You can import distributed property values from Classic Models
- Map elevation to a field in source data when importing points from shapefile
- Model outputs including groundwater heads, drawdown, velocity, and concentrations can be exported to text files
• Export contours as polyline shapefiles

**Usability and Other Enhancements**

• Head and Concentration Observations can be specified as separate groups so that you can view and filter these separately at the View Charts step
• Performance improvements on project open and close operations for larger projects.
• Output Control file translation supports the use of keywords or codes and the standard or compact budget formats to provide support for both MODPATH and RT3D.
• Added MT3D-MS and RT3D summary output (.OT) files to the open files dialog at the model run step

**Defects Addressed**

• Intermittent and unhelpful translation error “Vmac is running in the task bar, please close it and try again”.
• Deleting particles by drawing a box does not actually delete particles
• Well schedule not translated correctly if the project time unit is not in days
• In some cases, particularly if the whole output water table exceeds the model ground surface it was rendered at Elevation=1
• Modifications to a pumping well boundary condition propagate to all runs under the same grid
• Concentration observations contained incorrect column names
• Project Clean-up/Remove orphan objects from model
• If all boundary conditions of a given type were deleted, they would still run in the model due to advanced run settings. You are now prompted to remove the translated file or back-it up so that it no longer runs in the model
• In some cases, the schedule for pumping wells with relative times does not import correctly
• In some cases, assigning a schedule to a conceptual boundary condition results in an error message that the schedule occurs before the model start date even if this is not true
• Concentration observations incorrectly handled for simulations with multiple species
• Transient Fluxes (Model Results) are not shown or exported, only Flux from first time step.
• Fluxes not displaying properly in 3D Viewer
• Cells remain active if boundary condition is applied to ‘Top’ of model domain
• MODFLOW-USG, BCF package fails to translate if property values are edited
• When using a uniform grid, the CM->NM conversion does not add conceptual BCs assigned to model top to Layer 1 rather than the upper-most active layer
• When importing heads file as 3D gridded data to a Rotated Grid Project, error “Object reference not set to instance of object appears”
• Inconsistent/compound units are not correctly converted for certain RT3D reaction parameters
• Water table for USG does not take ponding into account
Reinstalling Visual MODFLOW Flex does not replace MODFLOW-SURFACT with the demo version in the engines folder.

Flex does not always cleanly delete elements from the Model Explorer, resulting in large file sizes.

In some cases, MODFLOW-USG conceptual model conversion results in incorrect parameter values, and causes parameter editing to fail.

Out of memory error when attempting to display certain DXF files in 2D view of Flex viewer or standalone 2D viewer.

Multiple Lake BCs result in a multiple key error upon translation.

For full version history, see the Visual MODFLOW Flex readme file at: https://www.waterloohydrogeologic.com/visual-modflow-flex-readme/

### 1.3.2 Version 5.1

#### Data Import/Export

- Surfaces created in Visual MODFLOW Flex can be exported as Surfer ASCII grid files (.GRD) for use in other Visual MODFLOW Flex projects and other software packages (e.g. Surfer, ArcMap, QGIS).

- Pumping Wells in Visual MODFLOW Flex numerical models can be imported as text files (.CSV) using the same format that they are exported from Flex projects allowing you to transfer Wells between projects.
Visualization and Settings

Vertical exaggeration in 3D views can be manipulated by keyboard commands:

The following keyboard commands adjust the vertical exaggeration in 3D-Views:

- `<Ctrl>Up>` - increase vertical exaggeration by 1
- `<Ctrl>Shift>Down>` - decrease vertical exaggeration by 10
- `<Ctrl>Up>` - increase vertical exaggeration by 1
- `<Ctrl>Shift>Down>` - decrease vertical exaggeration by 10
- `<Ctrl>Home>` - set to vertical exaggeration to 0
Packages, Properties, and Boundary Conditions

- Field values for Properties and Boundary Conditions are available as variables in the Expression Builder

Grid Editing

- You can now specify which numerical model run(s) are carried over when performing grid edits.
Defects Addressed

- In some scenarios (for instance if CHD cells are specified at the very bottom of the model [such as in the Henry model]), the parallel version of SEAWAT may have convergence problems that cannot be resolved by changing solver parameters.
- SEAWAT engine run output was truncated and formatted incorrectly.
- Concentration values of -1 in the specified concentration boundary condition (indicating the species should be skipped) were incorrectly translated to the SSM file as 0.
- Error in translating diffusion arrays to the DSP package when specifying the Multidiffusion option in MT3D-MS or SEAWAT runs.
- In rare occasions, property zones were set to -1 and uneditable.
- In some cases, successive model translation times increased due to a memory leak.
- In ZoneBudget runs, Flex does not read in values for zones that are not directly adjacent to Zone 1.
- Errors thrown if user skips straight to Translate without selecting an engine.
- In some cases, changes to stress periods do not update instantly to the translation settings resulting in incorrect translated time discretization and/or output control (.OC) files, following changes.
- In some cases, PEST results will not update correctly into a new run.
- An error is shown when user clicks [...] button to change the path to Previous MODFLOW Run HDS file if the HDS file is not present.
- Hitting Apply in the Settings dialog for a concentration doesn’t update the BC view properly.

For full version history, see the Visual MODFLOW Flex readme file at: https://www.waterloohydrogeologic.com/visual-modflow-flex-readme/

1.3.3 Version 5.0

SEAWAT

Visual MODFLOW Flex now supports SEAWAT (v4.00.04), a three-dimensional variable density flow and multi-species solute transport model developed by the USGS, based on a combination of MODFLOW and MT3D-MS. Version 4 of SEAWAT also supports viscosity effects and heat transport.
The following SEAWAT flow packages are supported in Visual MODFLOW Flex:

- **BAS6** - Basic Package
- **DIS** - Discretization Package
- **BCF6** - Block-Centered Flow Package
- **LPF** - Layer Property Flow Package
- **RCH** - Recharge Package
- **EVT** - Evapotranspiration Package
- **WEL** - Well Package
- **DRN** - Drain Package
- **RIV** - River Package
- **GHB** - General-Head Boundary Package
- **FHB** - Flow and Head Boundary Package
- **CHD** - Time-Variant Constant Head Package
- **LAK** - Lake Package (LAK3)
- **HFB6** - Horizontal Flow Barrier package (HFB6)
- **OBS** - Observation
- **HOB** - Hydraulic-Head Observation
- **GMG** - Geometric Multi-Grid Solver
- **PCG** - Pre-conditioned Conjugate Gradient Solver (PCG2)
- **WHS** - BiCGSTAB-P Matrix Solver

Source data courtesy of the USGS ([SJR2016-5022](https://scihub.copernicus.eu/dhus/landing?process=createFromIdentifier&identifier=SJR2016-5022))
The following SEAWAT transport packages are supported in Visual MODFLOW Flex:

- **VDF** - Variable Density Package
- **VSC** - Viscosity Package
- **BTN** - Basic Transport
- **ADV** – Advection
- **DSP** – Dispersion
- **SSM** - Source/Sink Mixing
- **RCT** – Reaction Package
- **GCG** - Generalized Conjugate Gradient Solver

**NOTE**: Advanced Translate/Run Settings (see below) enable partial support for simulating additional processes/packages (e.g. subsidence, drain return) developed outside of Flex.

Visual MODFLOW Flex now supports **RT3D** (v2.50), an advanced reactive transport engine based on the popular MT3DMS transport engine.

The following RT3D reaction modules are supported in Visual MODFLOW Flex:

- Instantaneous aerobic degradation of BTEX.
- Six Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors
- Rate-Limited Sorption
- Double Monod Model
- Sequential First-Order Decay (up to 4 species, e.g., PCE/TCE/DCE/VC)
- Aerobic/Anaerobic PCE/TCE Dechlorination

**Visualization and Settings**

- **Cell Inspector**: View and inspect structured and unstructured models at the cell-level using the new cell inspector, available in the workflow viewers. The cell inspector displays cell values for user-specified input and output data fields.

![Cell Inspector](image)

- **Status Bar**: Cell dimensions have been added to the status bar.

```
```

- **Time-Picker**: A time-picker is now available for unstructured grid workflow viewers.

![Time-Picker](image)

- **Layer-Picker**: A layer-picker is now available for unstructured grid workflow viewers.

![Layer-Picker](image)
- Workflows

- **Additional Translation/Run Settings:** Added advanced settings for all supported engines (excluding MODFLOW-USG) so that you can specify whether individual packages are translated and/or run with the model, which provides the flexibility to integrate your own package input files within the Flex environment.

- Property and Boundary Condition Tools

- **Field Calculator:** Populate boundary condition fields using a calculator that includes an expression builder dialog to facilitate the inclusion of variables based on your model’s properties (e.g. grid dimensions, elevations).
• **Dispersion Parameters**: Values can be edited in the Dispersion Parameter Dialog using copy, paste, and fill commands.

![Dispersion Parameters Dialog](image)

- **Usability and Other Enhancements**
  - **Easy Classic Importing**: Import your Classic project into Visual MODFLOW Flex directly from the file menu.
  - **Import Classic Concentrations**: Boundary condition concentrations are now imported from Classic Projects.
  - **Animate Relative times in 3D views**: in 3D View animations, the time stamp label for heads can be specified in relative or absolute time.
• **Descriptive Title Bars for Settings Windows**: settings windows now display descriptive window titles including the data layer and viewer the settings to which they apply.

• **Rename Objects**: objects in the Data Explorer and most objects in the Model Explorer can be renamed.

• **Feedback Button**: added anonymous feedback buttons so that you can easily report what features you like or don’t like.

• **SAMG Solver**: added a single-processor version of the SAMG solver for Visual MODFLOW PRO users.

---

**Defects Addressed**

• Defining BCs along irregular polylines that are close to the active model limits produced unexpected results

• Reorder Tool was missing in some views

• The Show/Hide Gridlines button and associated toolbar disappeared intermittently from the viewer

• Well Data import failed without warning the user that the pumping schedule had inconsistencies, including pumping start date before model start date or pumping start date after pumping end date

• User could not assign polygon attribute data to conceptual recharge/evapotranspiration elements

• Problem(s) with Time Series Viewer

• Multiple PEST Workflows could be created for the same model run which corrupted the PEST workflow.

• Multiple issues in the advanced run/translation settings

• PCG Solver translation settings of the PCG2, PCG4, and PCG5 solver packages for MODFLOW-SURFACT models

---

For full version history, see the Visual MODFLOW Flex readme file at: [https://www.waterloohydrogeologic.com/visual-modflow-flex-readme/](https://www.waterloohydrogeologic.com/visual-modflow-flex-readme/)

### 1.3.4  Version 4.1

---

**Visualization and Settings**

• **2D/3D View Keyboard Navigation**: Navigate in the active 2D or 3D view using the keyboard. Arrow keys pan the view, +/- keys (on the number pad) and page up/page...
down keys zoom in and out. In an active 3D view, the <shift>+arrow up/down keys changes the inclination of the model relative to the horizon and <shift>+arrow right/left rotates the model about the vertical axis.

- **Custom Contours**: Customize contours in 2D, 3D, and Flex Viewers by setting start and finish contour values for contour lines and isolines. You can also specify a contour interval. Changes in one part of the Flex Viewer are reflected in the other parts.

- **3D View Setting Defaults**: Create and Save Project-Specific Default View Settings for each each data type in the 3D Viewer.

- **Time-Series Viewer**: View time series data such as heads, saturation, and concentration at points within transient models without specified observation points.

### Workflows

- **Modeling Objectives**: Modeling Objectives for Transport Simulations can be changed at any time, not just during initial setup or conversion.

- **Model Settings for Translate/Run**: You can now specify whether individual packages are translated and/or run with the model, which provides the flexibility to use your own packages similar to the advance settings in Classic for model translate/run.

### Grid Editing Enhancements

- **Cell Size Control for Conceptual Models**: Similar to numerical grids, you now have full control over the grid cell size specification during initial grid creation; you can set grid cells to whole numbers for ease of calculation and reporting.

### Property and Boundary Conditions

- **Boundary Conditions in LGR Models**: Boundary conditions that span the parent/child grid interface(s) in an LGR model can be specified in both the Conceptual and Numerical workflows.

- **Dual Domain mass transfer**: Flex now supports dual domain mass transport modeling through the MT3DMS and MODFLOW SURFACT (see below) transport engines.

- **Unsaturated Zone Flow (UZF) Package**: Flex now supports 1-D unsaturated flow associated with the UZF package.
Updated Engines

- **Upgraded MODFLOW 2005**: Flex is now packaged with MODFLOW-2005 v1.12.00, the latest official USGS version as of the release date of Flex 4.1.

- **Upgraded MODFLOW NWT**: Flex is now packaged with MODFLOW-NWT v1.1.2, the latest official USGS version as of the release date of Flex 4.1

MODFLOW SURFACT

- **Added Support for MODFLOW SURFACT**: Visual MODFLOW Flex now supports MODFLOW SURFACT Version 4.0, a fully-coupled flow and transport engine developed by Hydrogeologic, Inc. which features many robust methods and enhanced simulation capabilities for handling complex saturated/unsaturated subsurface flow and transport processes. Based on the USGS modular groundwater flow model, MODFLOW, it supports the following packages:

<table>
<thead>
<tr>
<th>Package Type</th>
<th>Package</th>
<th>Description</th>
<th>Ext.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Flow and Transport</strong></td>
<td>BAS4</td>
<td>Basic Package</td>
<td>BAS</td>
</tr>
<tr>
<td><strong>Processes</strong></td>
<td>BCF4</td>
<td>Block-Centered Flow Package</td>
<td>BCF</td>
</tr>
<tr>
<td></td>
<td>BTN1</td>
<td>Basic Transport Package</td>
<td>BTN</td>
</tr>
<tr>
<td></td>
<td>TMP1</td>
<td>Time-Varying Material Properties Package</td>
<td>TMP</td>
</tr>
<tr>
<td><strong>Flow and Transport</strong></td>
<td>CHD</td>
<td>Time-Variant Specified Head</td>
<td>CHD</td>
</tr>
<tr>
<td><strong>Boundary Conditions</strong></td>
<td>DRN</td>
<td>Drain Package</td>
<td>DRN</td>
</tr>
<tr>
<td></td>
<td>EVT</td>
<td>Evapotranspiration Package</td>
<td>EVT</td>
</tr>
<tr>
<td></td>
<td>FHB</td>
<td>Flow and Head (Time-Varying) Boundary</td>
<td>FHB</td>
</tr>
<tr>
<td></td>
<td>FWL5</td>
<td>Multi-Node Fracture Well</td>
<td>FWL</td>
</tr>
<tr>
<td></td>
<td>GHB</td>
<td>General-Head Boundary</td>
<td>GHB</td>
</tr>
<tr>
<td></td>
<td>HCN1</td>
<td>Prescribed-Head-Concentration Boundary</td>
<td>HCN</td>
</tr>
<tr>
<td></td>
<td>HFB</td>
<td>Horizontal Flow Barrier (Wall)</td>
<td>HFB</td>
</tr>
<tr>
<td></td>
<td>LK3</td>
<td>Lake Package V3</td>
<td>LAK</td>
</tr>
<tr>
<td></td>
<td>PCN1</td>
<td>Prescribed-Head-Concentration Boundary</td>
<td>PCN</td>
</tr>
<tr>
<td></td>
<td>RSF4</td>
<td>Recharge-Seepage Face Package</td>
<td>RCH</td>
</tr>
<tr>
<td></td>
<td>RIV</td>
<td>River Package</td>
<td>RIV</td>
</tr>
<tr>
<td></td>
<td>WEL</td>
<td>Well Package</td>
<td>WEL</td>
</tr>
<tr>
<td><strong>Observations</strong></td>
<td>HOB</td>
<td>Head Observation Package</td>
<td>HOB</td>
</tr>
<tr>
<td></td>
<td>OBS</td>
<td>Observation Nodes</td>
<td>OBS</td>
</tr>
<tr>
<td><strong>Output Control</strong></td>
<td>OC</td>
<td>Output Control</td>
<td>OC</td>
</tr>
<tr>
<td><strong>Solvers</strong></td>
<td>PCG5</td>
<td>Preconditioned Conjugate Gradient Solver</td>
<td>PCG</td>
</tr>
<tr>
<td></td>
<td>SIP</td>
<td>Strongly Implicit Procedure</td>
<td>SIP</td>
</tr>
</tbody>
</table>
### NOTE

MODFLOW SURFACT was developed by Hydrogeologic, Inc. and is sold as a separate add-on for Visual MODFLOW Flex. A demo-version that supports model grids of up to 10,000 cells is included for evaluation purposes. Additional packages (e.g. subsidence [SUB], streamflow routing [SFR], etc.) are supported by SURFACT and may be developed outside Flex.

### Usability and Other Enhancements

- **Undock/redock Tabs**: Tabs can be undocked as separate windows. Simply drag and drop or right-click on the Tab name and select the "Undock Tab" option and you can place the new window anywhere on your monitor screen(s). Redock a window back to the main program window using drag and drop or right-click and select "Dock Tab" option.

- **Explorer Support**: Open a Visual MODFLOW Flex .AMD file directly from Windows Explorer.

- **.Net Framework 4.6.1**: Now bundled with Flex installation.

- **About/Splash Screen**: Version and Build number are displayed on the Splash Screen and on the About Screen.

- **Improved Performance**: Improved memory handling and performance enhancements.

### Defects Addressed

- Could not edit or define a Boundary Condition when multiple grids were present as part of the same LGR grid.

- Inconsistency between calculating and displaying leakance in the lake boundary condition when using certain time units.

- Some PEST runs were not successful due to incorrect observation files.

- Issue when importing time schedules for recharge package.

- Translation issue when importing some Visual MODFLOW 2010.1 Classic projects into Flex.

- Progress bar does not provide adequate feedback when opening large projects.

- Issue with extracting multiple species from sub-grids for MT3DMS runs.

- Toolbox was blank after translating/running the model and clicking Define Properties step.
• Translation issue in MT3DMS RC3 file for bulk density values when using inconsistent units
• Improved feedback/reporting for certain engine runs that do not converge

For full version history, see the Visual MODFLOW Flex readme file at:
https://www.waterloohydrogeologic.com/visual-modflow-flex-readme/

1.4 How to buy

You can buy Visual MODFLOW Flex directly through your local sales representative, via e-mail/phone, or online through the shopping cart on our website using a credit card.

Direct order link and Visual MODFLOW Flex Software homepage:

Contact Us:

www.waterloohydrogeologic.com

Find your local distributor:

www.waterloohydrogeologic.com/global-distributors/

E-mail a Sales Representative:

sales@waterloohydrogeologic.com
2 Program Overview

In order to become the most efficient and effective in the Visual MODFLOW Flex environment, it is recommended that you familiarize yourself with a few simple concepts, terminology, and where you can find and access things.

Data Objects

All of the data that you interact with in Visual MODFLOW Flex are referred to as data objects. These can consist of:

- Raw Data that you have:
  - Imported: From polyline or polygon shapefiles, wells from a spreadsheet, surfaces from Surfer .GRD, etc.
  - Created: Through digitizing points, polygon, or polylines
• Conceptual Data Objects: These are generated as you progress through the conceptual modeling workflow, and include:
  • Horizons, Structural Zones, Property Zones, and Boundary Conditions.

• Numerical Model Data Objects. These are generated as you progress through a numerical modeling workflow, and include:
  • Input: Numerical Grid, Properties (Conductivity, Initial Heads, etc..), Boundary Conditions (a group of river cells, drain cells, pumping well cells, etc..), Observation Wells, Zone Budget zones, and Particles.
  • Output: Calculated Heads, Drawdown, Pathlines, etc.

Each data object will have a check box beside it, allowing it to be displayed in different 2D/3D viewers. Data objects can be re-ordered, renamed, or grouped into data folders. To create a data folder, right-click in the Data are Object explorer and select New Folder. Data folders can be renamed; however, in the current version of Visual MODFLOW Flex they cannot be moved.

Each data object also has Settings which can be accessed by right-clicking on the data object in the tree, and selecting Settings. The settings provide access to general properties (statistics, file origin, etc.) and Style settings (symbol colors, shape, labeling, etc.) for those objects displayed in an active viewer. For more details, see Data Settings.

Many wizards and dialog boxes in Visual MODFLOW Flex require you to select data objects from the Data Explorer or Conceptual Model Explorer, e.g., when defining horizons, creating property zones, and assigning attributes to boundary conditions.

When you see a Blue Arrow located next to an input field in a dialog box or a wizard, this means that a data object selection is required. Simply click the appropriate data object from the Data Explorer or Conceptual Model Explorer and then click the Blue Arrow button to insert the data object into the input field.

== Model Explorer

The Model Explorer contains all of the conceptual models and numerical models, and corresponding data objects for your project. To help you locate items in the model explorer, we have included a search option which will automatically filter the items in the tree based on what you type.

Conceptual Model Tree  Numerical Model Tree
2D/3D Viewers

Data objects can be displayed in one or more of the following viewers:

- **2D View**: Plan view; ideal for GIS data, surfaces, well locations, images, etc.
- **3D View**: Ideal for data that have X,Y and Elevation (Z) values defined: such as Structural Zones, Wells, Pathlines, Heads along a cross-section, etc.
- **“Flex” View**: Ideal for complex views and available in the numerical modeling workflows, the Flex viewer consists of a combination of a Layer, Row, Column, and 3D sub-views. The individual sub-views can be shown/hidden.
Groundwater modeling consists of a series of steps that must be completed in a particular sequence in order to achieve a specific goal. In Visual MODFLOW Flex, these steps are presented in a workflow. In the Workflow window, you see the steps that make up a workflow and at each step there is a corresponding GUI with which you interact. The benefits to you as a modeler are unlimited:

- **Simplicity**: You know where you are and where you have to go. This dramatically reduces the learning curve.
- **Accessibility**: all the actions you need are available at your fingertips; no more hunting for an option deep inside a menu.
- **Convenience**: modeling is iterative and requires a frequent amount of flipping between input, run, and results. The workflow GUI simplifies these back-and-forth tasks.

In Visual MODFLOW Flex, there are separate workflows for Numerical Modeling, for Conceptual Modeling, and for PEST (Parameter Estimation/Sensitivity Analysis).

The workflow panel contains a toolbar and a list of steps required for your current workflow.

**Navigating a Workflow**

- Go to the Previous Step in the workflow
- Go to the Next Step in the workflow
Workflow States
Beside each state in the workflow there is a corresponding icon. The icon helps you to identify which is your current step, which steps have been completed, and which steps you may proceed to next. The image below provides an explanation of this.

File Structure
Visual MODFLOW Flex projects consist of a project file and associated folders with the same base name. For example, if you have a Visual MODFLOW Flex project called "Model", the entire project consists of the file "Model.amd" and the folder (and its contents) in the same path location called "Model.data".

In order to transfer or manually backup a Visual MODFLOW project from one file location to another, you will need to copy both the .AMD file and its associated folder to the new location.
2.1 Main Menus and Toolbars

The following sections describe the various menu and toolbar options in Visual MODFLOW Flex.

<table>
<thead>
<tr>
<th>Menu Items</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>File Menu</strong></td>
</tr>
<tr>
<td>The File menu provides access to standard operations such as open, save, save as, and close project. The file Menu can also be used to import data from external sources and modify project settings. You will find the last few models you have opened so you can quickly re-open them.</td>
</tr>
</tbody>
</table>

**File Operations:**

- **New Project <CTRL+N>:** Opens the create new project setup dialog.
- **Open Project <CRTL+O>:** Opens an existing project. Note: only one project can be open in Flex at a time.
- **Import Project <CTRL+I>:** Imports a Visual MODFLOW Classic (*.vmf) file.
- **Close:** Closes the active project

**Project Operations:**

- **Save Project <CTRL+S>:** Saves the active project to its current location.
- **Save Project As <F12>:** Saves the active project in a selected location.
- **Project Settings:*** Opens the Project Settings dialog, which stores model setup and project meta-data information: including model default values, units and coordinate system.
- **Import Data <CRTLD-D>:** Opens the Import Data dialog to add external data to your project.
- **Recent Project List:** A list of the most recently saved projects will be shown. Selecting any of the projects in the list will close any active project (upon user confirmation) and open the selected project.
- **Exit <Alt+F4>:** Exits Flex - you will be prompted to save your project if one is open.
**View Menu**
The View menu allows you to show/hide the Data and Model Explorer <F11> and the Workflow tree <F4>.

**Tools Menu**
The Tools menu provides the following options:

- **Free Memory**: allows you to free up the memory usage if you have been running a number of high-demand, 3D visualization operations

- **Preferences**: opens the Preferences settings dialog which allows you to adjust the settings for 3D Viewers and for Flex in general.

**Restore System Settings**
Use this option to reset Visual MODFLOW Flex to the factory style settings.

**Project Color Palette**
Visual MODFLOW Flex provides an option to use Project-wide Color Palettes. This is useful when you have multiple data objects that are rendering the same attribute (eg. heads from different model runs, conductivity distributions), and you want to make qualitative comparisons between these. This is challenging when each data object has its own min and max values and are colored based on this. However, it becomes much easier when these data objects all read from a common color palette.
The Project Color Palette settings allow you to set the min/max values for various attributes; these can then be used for various data objects and viewers throughout the project.

To enable the Project Color Palette option for a specific data object, this must be enabled in the Color by Attribute page; see Color by Attribute for more details.

**Workflow Menu**

Allows you to create a new Numerical Model workflow or Conceptual Model workflow or open existing workflows. When you select the appropriate item, the associated workflow window will load.

**Window Menu**

Allows you to create a new 2D, 3D, or time series viewer of the active data object selected in the Data Viewer/Model Explorer or reset the active view to the default extents.

**Help Menu**

Provides links to help topics, webhelp, and online resources. The Just-In-Time Help will display a small help panel below most steps in the workflow window.

- **Help Topics (F1)**: opens the built-in help documentation including tutorials and getting started guides
- **Web Help**: opens the online-help documentation
- **Release Notes**: opens the online documentation for the changes associated with each version
- **Online Support and Resources**: opens the online page for accessing technical resources and contact information for technical support
- **Checks for Updates**: calls out to Waterloo Hydrogeologic’s webpage to see if updates are available.
- **Show Error Log**: opens the Visual MODFLOW error log, which can be useful in helping technical support diagnose and pinpoint problems, if they arise.
Show System Information: opens a text file with system configuration information, which can be useful in helping technical support diagnose problems, if they arise.

About... launches the About Screen, which shows the version and build you are using along with your registration details (name, company, serial number).

Licensing Help will load the online help for working with licensing. For more details on licensing, please refer to the Getting Started Guide.

Toolbars

Below the main menu, there are two toolbars:

The toolbar in the first row contains application/project level options; from left to right, these options are:

- New Project
- Open Project
- Save Project
- Show/Hide the Model Explorer and Data tab
- Show/Hide the Workflow Tree
- Help
- Thumbs Up/Down - for anonymous feedback and product feature requests

The toolbar in the second row contains options for adjusting the Data tab and Model Explorer; from left to right, these options are:

- Combine Data tab and Model Explorer into one view, separated by tabs
- Separate the Data tab and Model Explorer into one frame above the other (factory default setting)
- Collapse All (levels of the tree)
- Collapse to Selected item on the Model Explorer
- Dock Model Explorer to the left side of the window
- Dock Model Explorer to the right side of the window

2.1.1 Preferences

The preferences menu allows you to adjust various settings in Visual MODFLOW Flex:

- Flex Viewer
• 3D Viewer
• General

Flex Viewer Settings

The Flex Viewer Settings include the option to select which properties are shown in the status bar and how they are formatted.

![Preferences dialog box showing Flex Viewer settings for 3D and General views.]

3D Viewer Settings

The following settings are available for 3D Views in Flex:
OpenGL Driver

By default, Visual MODFLOW Flex will attempt to use the vendor-provided driver included with your graphics acceleration hardware. If problems are encountered with the vendor-provided drivers, e.g., poor on-screen display/performance, Visual MODFLOW Flex provides the option to use the Microsoft Driver for OpenGL.

Virtual Data Grid

Depending on the size of your model, Visual MODFLOW Flex may run very slowly during rotations or when 3D Numerical Gridded data objects are moved in the 3D Viewer (e.g., MODFLOW Heads, Drawdown, Properties, or Transport Concentrations). In this situation, the virtual data grid option may be used to increase the speed of the data processing and image rendering. It can be used to set up a uniformly spaced grid with a specified number of rows and columns.

The virtual data grid option will interpolate the data from the model to the uniformly spaced virtual grid. This allows a smaller amount of information to be processed much faster. However, this also results in a loss of resolution of the data, and some local scale minimum and maximum values may be missed.
If you are experiencing performance issues, try choosing the "Low" option. The number presented in parentheses is the number of cells used along the X and Y axis for the virtual data grid.

**Surface Visualization Quality**

This parameter controls the visualization quality and speed of surface data objects for the 3D viewer. This parameter can be used when you want to display surfaces with a large grid size (eg. 500 x 500 or greater). Similar to the Virtual Data Grid option above, Surfaces will be resampled to the defined grid size (where the number in parentheses represents the uniform number of rows/columns).

---

**Surface and Horizon Quality**

The Surface Visualization Quality parameter will affect 3D visualization only; the resolution of the surface data object will not be adjusted, and will be honored when using this surface for Horizons or Property values in a numerical model.

Horizon quality is defined by quality of surface used for horizon. The maximum grid size for horizons is 1,000 rows x 1,000 columns. In the case where you select a Surface that has a higher resolution than 1,000 x 1,000, Visual MODFLOW Flex will coarsen that surface to a maximum horizon quality of 1,000 x 1,000. This grid size can be increased; however there will be a performance loss when doing a Conceptual to Numerical conversion and 3D visualization of Horizons and Conceptual Model structural zones and property zones. For more details, please contact Technical Support.

---

**Point Style**

This setting provides two options for displaying points in 3D Viewer: Basic and Advanced. If the Basic option is selected, 3D Viewer will render the point shapes in the 3D Viewer. On some computers this option may hinder the performance of the 3D Viewer. If the Advanced option is selected, 3D Viewer will use bitmap images to display the points. If you are experiencing performance issues display points in 3D Viewer, the Advanced option should be selected.

**Please Note:** The Basic option only supports cube and sphere symbols for displaying points.
General Settings

By default, Visual MODFLOW Flex collects anonymous usage statistics. The data that we collect enables us (Waterloo Hydrogeologic) to get reliable statistics on which features are used (or not used). This helps us better understand the needs of our users so we can build better applications. The information collected is completely anonymous – it is not traceable to any individual, project, or organization.

You can opt out by deselecting this option. For more information about our data collection, please see here.

![](preferences.png)

2.1.2 Project Settings

The project settings dialog allows you to adjust various project-specific settings in Visual MODFLOW Flex:

- **Project Information Tab**
- **Project Coordinates**
- **Units**
- **Model Defaults**

Project Information Tab

The project information tab includes the following information:

- **Name**: The filename of the project
- **Data Repository**: The file path of the project
- **Description**: A long form description of the project
Project Coordinate System

The Project Coordinate tab contains a description of the project spatial reference system, which includes:

- **Projection**: a systematic transformation of latitudes and longitudes of locations from the surface of a sphere or ellipsoid onto a two-dimensional plane. Supported examples include the Universal Transverse Mercator (UTM) Projection, and Lambert conformal conic (LCC) (used in the State Plane spatial reference system).

- **Datum**: The horizontal datum is the model used to measure positions on the Earth. A specific point on the Earth can have substantially different coordinates, depending on the datum used to make the measurement. There are hundreds of local horizontal datums around the world, usually referenced to some convenient local reference point. Contemporary datums, based on increasingly accurate measurements of the shape of the Earth, are intended to cover larger areas. The WGS84 datum, which is almost identical to the NAD83 datum used in North America and the ETRS89 datum used in Europe, is a common standard datum. Supported examples include WGS84, NAD27, NAD83, and ETRS89.

- **Units**: Units of length and distance. Supported examples include feet and meters.
Please Note: The project spatial reference is set when the project is created and cannot be subsequently changed. Shapefiles with a supported specified spatial reference system other than the project spatial reference system will be automatically transformed to the project spatial reference system during the import process. If the project spatial reference system is unknown or in a local or unsupported coordinate system, use the Local Cartesian spatial reference system; however, all imported data is assumed to be in the same local spatial reference system and no transformations will be applied to imported shapefiles in this case.

Units

Visual MODFLOW Flex allows you to specify which measurement units are used in your project. Measurement units are only used during model translation to convert all units into consistent base units.

Available units include the more common units in the International System of (SI) Units, informally known as the metric system, and the United States Customary System (USC) system. Please note that the imperial system of measurements is closely related to the USC, but there are differences (e.g. 1.0 imperial gallons ≈ 1.200 945 US liquid gallons).
Consistent Units

All of the modeling engines (e.g. MODFLOW-2000, -2005, -NWT, -LGR, -USG, -SURFACT, SEAWAT, MT3D-MS, and RT3D) rely on consistent units which are specified using the three major base units associated with groundwater flow, transport, and related processes - length (L), mass (M), and time (T). Note that the temperature units (K), which are used in heat transport simulations supported by the SEAWAT engine, are hard-coded to use degrees Celsius.

Compound Units

Compound measurement units are those measurement units that consist of combinations of base units. Examples include conductivity (L/T), recharge (L/T), concentration (M/L³), and specific storage (1/L). As discussed above, compound units are converted into consistent base units at the time of model translation. That is all compound units are converted from the specified compound units into units that are consistent with the individual base units. For example, in a project with base units of meters for length and days for time, a recharge value of 3.65 mm/yr would be converted into 1.0x10⁻⁵ m/d at translation. Note that output values with compound units (e.g. concentrations) are converted back into the specified project units when being displayed in Visual MODFLOW Flex.

⚠️ Please Note: In Visual MODFLOW Flex, changing from project unit to another will not change the magnitude of the value shown in the user interface. For example, if you change the conductivity units from ft/d to m/d, a value of 1 ft/d would be 1 m/d in the model run conductivity arrays rather than 0.3048 m/d.
## Units Supported in Visual MODFLOW Flex

<table>
<thead>
<tr>
<th>Measurement</th>
<th>Dimension</th>
<th>Supported SI Units</th>
<th>Supported USC Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concentration</td>
<td>M/L³</td>
<td>• mg/L</td>
<td>• ft/s</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• µg/L</td>
<td>• ft/d</td>
</tr>
<tr>
<td>Conductivity</td>
<td>L/T</td>
<td>• m/s</td>
<td>• cm/s</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• m/d</td>
<td>• cm/d</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ft/s</td>
<td>• ft/d</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ft/min</td>
<td>• ft/min</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ft/s</td>
<td>• ft/s</td>
</tr>
<tr>
<td>Density</td>
<td>M/L³</td>
<td>• g/cm³</td>
<td>• ft/s</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• g/m³</td>
<td>• ft/d</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• kg/m³</td>
<td>• ft/d</td>
</tr>
<tr>
<td>Length*</td>
<td>L</td>
<td>• m</td>
<td>• ft</td>
</tr>
<tr>
<td>Mass*</td>
<td>M</td>
<td>• kg</td>
<td>• ft</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• g</td>
<td></td>
</tr>
<tr>
<td>Pumping Rate</td>
<td>L³/T</td>
<td>• m³/d</td>
<td>• ft/d</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• m³/min</td>
<td>• ft/d</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• m³/s</td>
<td>• ft/d</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ft/d</td>
<td>• GPM</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• ft/yr</td>
<td>• GPD</td>
</tr>
<tr>
<td>Recharge</td>
<td>L/T</td>
<td>• m/yr</td>
<td>• ft/yr</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• m/d</td>
<td>• ft/yr</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• mm/yr</td>
<td>• ft/yr</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• mm/d</td>
<td>• ft/d</td>
</tr>
<tr>
<td>Specific Storage</td>
<td>1/L</td>
<td>• 1/m</td>
<td>• in/yr</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• 1/cm</td>
<td>• in/yr</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• in/d</td>
</tr>
</tbody>
</table>
**Measurement** | **Dimension** | **Supported SI Units** | **Supported USC Units**
--- | --- | --- | ---
Temperature* | K | • °C |  
Time* | T | • s | • d • yr

**Note:**
* Denotes a base unit

**Abreviations:**

- m = meter
- cm = centimeter
- mm = millimeter
- ft = USC survey foot

- µg = microgram
- mg = milligram
- g = gram
- kg = kilogram

- L = liter
- GPM = USC liquid gallons per minute
- GPD = USC liquid gallons per day

- s = second
- min = minute
- d = day
- yr = year

- °C = degree Celsius

---

**Model Defaults**

Model defaults are those values that will be populated into the relevant model properties on creation if no other value is specified in the development process. In other words, changing the model default values after model values have been created will have no effect on the values in the model as these values have already been applied. Certain physical constants/material properties are also specified at this tab.
2.2 Project Version Compatibility

Visual MODFLOW Flex is backwards compatible with projects made in earlier versions. However, Visual MODFLOW Flex is not forwards compatible, which means you cannot open a project that was made in a newer version of the software than you are running. This generally means that if you are working on modeling projects in a team environment, it is recommended that all members of the team work using the same version of Visual MODFLOW Flex.
Creating a full backup is strongly recommended before opening projects made in older versions of Flex, since Visual MODFLOW may update certain files during common operations without an explicit save. Please use caution when opening projects made in older versions of Flex without such a backup.

**Working with older projects in Flex**

From Visual MODFLOW Flex version 6.1 and onwards, the application will detect if you are using the latest version and prompt you to back up your project automatically.

**Backup Project**

Project backups will include a zip archive of the Visual MODFLOW Flex project files which consist of the `{project_name}.amd` and `{project_name}.data` folder. You may also optionally include the engine files and/or companion files.

**Engine files**: the model input and output files associated with individual engine runs (e.g. MODFLOW-2005, MT3D-MS, ZoneBudget, MODPATH, PEST, etc.). These files can be quite large and are often easily regenerated (if you developed the entire model in Visual MODFLOW Flex), so it may be preferable to exclude these.

- **Companion files**: external files such as basemap images, DXFs, shapefiles, associated with the project but that are stored outside of the project.data folder.

**See Also**

- Working with [legacy well objects](#) in projects prior to Visual MODFLOW Flex v2015.1
2.3 Keyboard Shortcuts

The following keyboard shortcuts are available when using Visual MODFLOW Flex:

- **General Shortcuts**
- **3D-Views**
- **Translation Step - Translated Package Tabs**

**General Shortcuts**

The following shortcuts are applicable when the main window is open and active:

<table>
<thead>
<tr>
<th>Command</th>
<th>Shortcut</th>
<th>Description/Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Help Topics</td>
<td>F1</td>
<td>Opens the built-in help documentation including tutorials and getting started guides</td>
</tr>
<tr>
<td>New Project</td>
<td>CTRL N</td>
<td>Opens the Create New Project setup dialog</td>
</tr>
<tr>
<td>Open Project</td>
<td>CTRL O</td>
<td>Opens an existing project</td>
</tr>
<tr>
<td>Import Project</td>
<td>CTRL I</td>
<td>Imports a Visual MODFLOW Classic (*.vmf) file</td>
</tr>
<tr>
<td>Save Project</td>
<td>CTRL S</td>
<td>Saves the active project to its current location; except at the Translation workflow step where the shortcut saves the active package file</td>
</tr>
<tr>
<td>Save Project As</td>
<td>F12</td>
<td>Saves the active project in a selected location</td>
</tr>
<tr>
<td>Import Data</td>
<td>CTRL D</td>
<td>Opens the Import Data dialog to add external data to your project</td>
</tr>
<tr>
<td>Exit</td>
<td>ALT F4</td>
<td>Exits Flex - you will be prompted to save your project if one is open</td>
</tr>
<tr>
<td>Workflow Tree</td>
<td>F4</td>
<td>Shows/hides the workflow tree for the active workflow</td>
</tr>
<tr>
<td>Model Explorer</td>
<td>F11</td>
<td>Shows/hides the model explorer/data explorer panel</td>
</tr>
</tbody>
</table>

**3D-Views**

The following shortcuts are applicable when a 3D-Viewer is active (i.e. its title bar is shown in blue)

<table>
<thead>
<tr>
<th>Command</th>
<th>Shortcut</th>
<th>Description/Result</th>
</tr>
</thead>
</table>

© 2019 by Waterloo Hydrogeologic
Shift Model up  
Shifts the model up in the viewer relative to the viewer window

Shift Model down  
Shifts the model down in the viewer relative to the viewer window

Shift Model left  
Shifts the model left in the viewer relative to the viewer window

Shift Model right  
Shifts the model right in the viewer relative to the viewer window

Reset Model View  
Re-centers the 3D View to the model centroid and the view/rotation angle to directly above the model and aligned North

Rotate towards horizon  
Rotates the foreground of the model upwards about its central axis parallel to the horizon

Rotate model away from horizon  
Rotates the foreground of the model downwards about its central axis parallel to the horizon

Shift Model left  
Rotates the model counter-clockwise about its central vertical axis perpendicular to the horizon

Shift Model right  
Rotates the model clockwise about its central vertical axis perpendicular to the horizon

Increase Vertical Exaggeration  
Increases the vertical exaggeration of the view by 1

Greatly Increase Vertical Exaggeration  
Increases the vertical exaggeration of the view by 10

Decrease Vertical Exaggeration  
Decreases the vertical exaggeration of the view by 1

Greatly Decrease Vertical Exaggeration  
Decreases the vertical exaggeration of the view by 10

Reset Vertical Exaggeration  
Sets the vertical exaggeration of the view to 0

Translation Step - Translated Package Tabs

The following shortcuts are applicable at when a translated package tab is active at the Translation Workflow Step:
<table>
<thead>
<tr>
<th>Command</th>
<th>Shortcut</th>
<th>Description/Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Save Edits</td>
<td>CTRL S</td>
<td>Saves text edits to the active translated file</td>
</tr>
</tbody>
</table>
3 Quick Start Tutorials

The following tutorials provide a brief introduction on how to use Visual MODFLOW Flex. The objective is not to teach you every detail, but to familiarize you with basic principles and the way the program works. The steps are intentionally kept brief so that you can actually start using the program as quickly as possible. You are encouraged to explore the more detailed sections of the Help documentation to further familiarize yourself.

In Visual MODFLOW Flex, there are two workflows you can follow: Conceptual or Numerical Modeling. Please take a moment to review the summary below to help you decide where you should start.

Conceptual Modeling (Recommended for Creating New Models)

Use this option if you want to:

- Start a new modeling project and have various data types/formats (GIS etc.) for defining the geological layering, material properties, and boundary conditions;
- Simulate domains with complex geological layering (pinchouts and discontinuities);
- Evaluate multiple numerical grids for your project; or
- Work with unstructured grids

The Conceptual Modeling tutorial will walk you through the following steps:

- Loading your raw data
- Defining the Geological Structure
- Defining the Properties and Boundaries
- Defining a Grid or Mesh
- Converting this to a Numerical Model

Start the Tutorial!

Numerical Modeling (For Existing Models)

Use this option if you want to:

- Create a MODFLOW-based numerical model (define the numerical grid and populate the grid the properties and boundary conditions; similar to conventional Visual MODFLOW);
- Import a Visual MODFLOW project set (*.VMF); or
- Import a standard USGS MODFLOW data set (MODFLOW-2000, -2005)

Start the Tutorial!
See Also:
Several sample projects are available for download from our tutorial webpage, that illustrate both the conceptual and numerical modeling workflows.

3.1 Conceptual Modeling Tutorial

The following example is a quick walk-through of the basics of building a conceptual model and converting this to a numerical model.

Objectives

- Learn how to create a project and import your raw data
- Become familiar with navigating the GUI and steps for conceptual modeling
- Learn how to define a 3D geological model and flow properties
- Define boundary conditions using your GIS data
- Define a MODFLOW grid, then populate this grid with data from the conceptual model
- View the resulting properties and boundary conditions
- Translate the model inputs into MODFLOW packages and run the MODFLOW engines
- Understand the results by interpreting heads and drawdown in several views
- Check the quality of the model by comparing observed heads to calculated heads

Required Files
Several files are required for this exercise, which should be included with the Visual MODFLOW Flex installation.

These files are available in your public "My Documents" folder, typically:

   C:\Users\Public\Documents\VMODFlex\Tutorials\Conceptual Model\supp files

If you cannot find these files, please download them from our website.

Creating the Project

- Launch Visual MODFLOW Flex 🎨.
- Select [File] then [New Project...]. The Create Project dialog will appear.
- Type in project name 'Conceptual Modeling Tutorial'.
- Click the [ ] button in Data Repository field, navigate to a folder where you wish your projects to be saved, and click [OK].
- Click the 'Create a folder for the project' checkbox
- Define your coordinate system and datum (or leave the default value - Local Cartesian).
- For this project, the default units will be fine.

The Create Project dialog should now look like this (ensure the units are the same):

- Click [OK]. The workflow selection screen will appear.
Select [Conceptual Modeling] and the Conceptual Modeling workflow will load.
In the Define Modeling Objectives step, you define the objectives of your model and the default parameters.
The **Start Date** of the model corresponds to the beginning of the simulation time period. It is important to define a relevant start date since your field measurements (observed heads and pumping schedules) will be defined with absolute date measurements, and must lie within the simulation time period. For this scenario, the default objectives will be fine.
Start Date

The start date will be used to retrieve pumping well and head/concentration observation data for the model run. When you define well data with absolute (calendar) dates, it is important that your start date reflects the actual start time for the model run. The well data must fall on or after that start date. Otherwise, these data will not be included in the simulation.

Also the start date cannot be changed once it has been set. If you inadvertently set the wrong start date, you can import your pumping well data and observation data in relative times (eg. starting at 0), and you will see no difference in the numerical model inputs/outputs.

Setting the Start Date
The model start date for this exercise should be set to 1/1/2000. Visual MODFLOW Flex uses a standard Windows date picker; a few tips are shown below on how to use this. Click on the button shown below, to load the Windows date picker.
The standard Windows calendar will appear. Click on the month in the header (as shown below):

All months for the current year will appear as shown below. Click on the year in the header:
A range of years will then appear as shown below. Click on the range of years in the header:

![Calendar with range 2010-2019 highlighted]

A list of years for the previous decade will appear. You can then use the < or > buttons to change the year. Select the 2000-2009 period:

![Calendar with range 2000-2009 highlighted]

Once you have reached the desired decade, select '2000' on the calendar as shown below:
A list of months will then appear for that year. Select January for this example, as shown below:

Finally, select "1" from the calendar as shown below:

The selected date (January 1, 2000) will then appear for the Start Date.

- For this scenario, the remainder of the default objectives and values will be fine.
- Click [Next Step] (Next Step) to proceed.
Collect Data Objects

- The next step is to import or create the data objects you will use for building the conceptual model.

At this step, you can import data, create new data objects (by digitizing) or create surfaces (from points data objects)

- Click the [Import Data] button and the following screen will load:
Select Polygon in Data Type combo box.
In the Source File field click the [...] button and navigate to your 'My Documents' folder, then 'VMODFlex\Tutorials\ConceptualModel\supp files', and select boundary.shp.

**Please Note:** The files may be located in the public documents folder: "C:Users\Public\Documents\VMODFlex\Tutorials\Conceptual Model", if you selected to make the program available to all users on install.

- Click [Next>>].
- Click [Next>>].
- Click [Next>>] then Click [Finish].
- The next step is to import a surface that represents ground surface.
- Click the [Import Data] button.
- Select Surface for the Data type.
- In the Source File field click the [...] button and navigate to the 'My Documents' folder, then 'VMODFlex\Tutorials\ConceptualModel\supp files' folder, and select ground.grd.
- Click [Next>>] through all the screens to accept the defaults, then click [Finish].
- Repeat these steps to import the remaining Surfaces: layer2-top.grd, layer2-bottom.grd.
- Next, import polyline data objects, and from the same source directory, select chd-east.shp; use all the defaults and finish the import.
- Repeat these steps, for polylines, importing first chd-west.shp, then river.shp.
- Once the data objects are imported, they will appear in the tree on the left side of the program window.
You can view these data objects in 2D or 3D; simply create a new viewer:

- Click on **Window** then **New 3D Window** from the main menu; an empty 3D Viewer will appear;
- Click on the check box beside each of the data objects you imported, and they will appear in the 3D Viewer.
- Increase the vertical exaggeration and reorient the screen so that you can see all your data objects, as in the image below.

On the Conceptual Model tab click **(Next Step)** to proceed, where you will arrive at the Define Conceptual Model step.
Define Conceptual Model

- Provide a name for the conceptual model (e.g. Conceptual Model 1), and model area.
- From the Data Explorer, select the polygon data object called boundary. This represents the conceptual model horizontal boundary, and then click the button to insert it as the reference object in the Define Conceptual Model workflow step.

**Please Note**: The model area cannot be defined using a complex polygon, or one that contains multiple polygons. A complex polygon is a polygon that intersects with itself.

- Click [Save]. Notice that the conceptual model elements have been added to the Model Explorer tree in the lower left section of the application window.

- Click [Next Step] to proceed to the Define Model Structure step.
Define Structure

Defining the geological model consists of providing geological surfaces as inputs for horizons. Then three-dimensional solids are created between these horizons. To create new horizons, follow the steps below.

- From the Horizons Settings dialog (shown below), click the [Add Horizon] button to add a new horizon row to the Horizon Information table.
- Repeat this two more times so there are 3 new rows on the Horizons table.
- From the Data Explorer, select the ground surface data object that will be used to generate the horizon.
- Click the button in Row 1 of the Horizons grid, to insert it into the Horizon Information table. See the example below.

- For this example, the default horizon type will be adequate. For information on each horizon type, please refer to the "Horizon Types" section of the user manual.
- Repeat the steps above to add additional horizons:
  - From the Data Explorer, select the layer2-top surface data object, click the button in Row 2 of the Horizons grid, to insert it into the grid.
  - From the Data Explorer, select the layer2-bottom surface data object, click the button in the Row 3 of Horizons grid, to insert it into the grid.

**Please Note:** Horizons must be added from the topmost geological layers and working downwards.

- You can preview the horizons in the adjacent 3D Viewer by clicking the [Preview Horizon] button.
- Once finished, you should see a display similar to the one shown below.
Click the [Create and Save] button to generate the model horizons, you should see them populate into the Model Explorer.

Finally, click the [ ] button to proceed to the next step (clicking the [ ] will also automatically generate the model horizons, if you didn't already click the [Create and Save] button).

**Define Property Zones**

Once you have imported sufficient raw data into your project, you can begin to construct one or more conceptual models using imported or digitized data objects as building blocks.

At this step, you can view/edit the flow properties for the model. There are two ways to define property zones: Using Structural Zones, or Using Polygon Data Objects.
Using Structural Zone(s)

This method allows you to create a property zone from existing structural zones in your conceptual model, i.e., zones generated from horizons.

- Click on the [Use Structural Zone] button as shown below
Select Zone1 structural zone from the conceptual model tree (under the Structure/Zones node as shown below)

Click the [ ] button to insert the zone in the Structural Zones field, as shown below.
Select the **Group of parameters** that will be defined, e.g., Conductivity, Storage or Initial heads. The data input grid below will display the appropriate parameters based on which parameter group is selected. For example, if conductivity is selected, the data input grid will show the parameters \( K_x, K_y, \) and \( K_z \). The data input grid will already be populated with the default values specified in the Project Settings ([File] > [Project Settings...]).

Type the desired values for the property zone. \( (K_x = 4\times10^{-6}, K_y = 4\times10^{-6}, K_z = 4\times10^{-7}) \)

Click on the [Save] button located on the right side of the window.

Repeat these steps for the other property zone:

- Click on the [Use Structural Zone] button
- Select Zone2 from the model tree
- Click on \( \rightarrow \) button to insert the zone in the Structural Zones field, as shown below.

Type the desired values for property zone2: \( (K_x = 7\times10^{-5}, K_y = 7\times10^{-5}, K_z = 7\times10^{-6}) \)

Click on the [Save] button located on the right side of the window.
Property zones can also be defined using polygon shapes; the values can also be defined from shapefile attributes or 2D Surface (distributed values). For more details, please see Defining Property Zones.

Click [Next Step] (Next Step) to proceed to the Selection screen.

In this screen, you can choose to proceed to Defining Boundary Conditions or proceed to Defining a grid or mesh.

Click the [Define Boundary Conditions] button to proceed.
In this window, you can choose the type of Boundary Conditions: Standard MODFLOW Boundary Conditions (CHD, DRN, RCH, etc.), Pumping Wells, or Walls.

- Click on the [Define Boundary Conditions] button:
The Define Boundary Condition dialog box will appear on your screen as explained in the following section.

**Define Boundary Conditions**

- At this step, you can define flow boundaries for the model.
- From the **Select Boundary Condition Type** combo box, select the desired boundary condition type.
- Select 'Constant Head'
- **Type** name: 'Constant Head East'.
- From the Data Explorer, select the **chd-east** polyline that represents this constant head.
- Click the button in the Define Boundary Condition dialog, to add this polyline to the input.
Click the [Next] button.

The next dialog allows us to define the constant head value. Visual MODFLOW Flex provides various options for defining boundary condition attributes. Attributes can be assigned from those stored in Surface, Time Schedule, Shapefile and 3D Gridded data objects. You can also set attributes as Static (no change over time) or Transient (changes over time).

For this tutorial, you will assign a static constant head value.
• In the empty fields located below the 'Starting Head (m)' and 'Ending Head (m)' fields type '347'.
• [Finish] button

Repeat these steps to define the other constant head boundary condition:

• Click on Define Boundary Conditions directly in the workflow tree
• Select the [Define Boundary Conditions] button.
• Choose Constant Head, select the chd-west polyline, and define a value of 325 for both the Starting Head and Ending Head
• Click [Finish].

Before you proceed, you will define one more boundary condition, a River.

• Click on Define Boundary Conditions in the tree, and select the [Define Boundary Condition] button.
• Choose River for the boundary condition type
• From the Data Explorer, select the river polyline
• Click the [ ] button in the Define Boundary Condition dialog, to add this polyline to the input.
• A warning may appear about clipping the polyline; click [OK] to continue
• Click the [Next>>] button.
• Define the following attributes for the river, as shown below: Stage = 335 (m), Bottom = 333 (m), Riverbed Thickness = 1 (m), Width = 10 (m), Riverbed conductivity = 0.01 (m/s).

• Click [Finish].
The River conceptual boundary condition will be added to the model tree.
The following display will appear.

Next you can choose what kind of grid to create:

- **Define Finite Difference Grid**: for a MODFLOW-2000, -2005, or MODFLOW-LGR model run;
- **Define Finite Element Mesh**: for preparing inputs for a FEFLOW .FEM file;
- **Define Unstructured V-Grid**: for a MODFLOW-USG run with a Voronoi grid; or
- **Define Unstructured Q-Grid**: for a MODFLOW-USG run with a QuadTree grid.

Click the **Define Finite Difference Grid** button and the following window will appear; define the inputs as explained in the following section.
Define Finite Difference Grid

- Enter a unique Name for the numerical grid. This name will appear in the Conceptual Model tree once the grid is created.
- Enter the grid size, and optionally, the grid rotation. The grid can be rotated counterclockwise about the grid origin by entering a value between 0 and 360 in the Rotation text field.
- The Xmin and Ymin values refer to the X-Y coordinates of the bottom-left corner of the numerical grid. The Xmax and Ymax values refer to the X-Y coordinates of the top-right corner of the numerical grid.
- The Columns and Rows fields allow you to define the Grid Size.
- **Type ‘100’** for both the # rows and columns
- Row and Column height/width will be automatically resized based on the grid extents and number of rows/columns, as shown below:
Click the [Next>>] button to proceed to define the vertical discretization. You will then see a cross-section preview of the grid.

By default, the vertical exaggeration is 1. Locate the 'Exaggeration' value below the preview window, and type '40' for the exaggeration value, then click [Enter] on your keyboard.
In the 'Define Vertical Grid' screen, specify the type of vertical discretization. For this exercise, the default Deformed grid be used. More details on the grid types can be found in the "Defining Grids/Meshes" section of the manual. Leave the defaults as is; click the [Finish] button. The Grid will then appear as shown in the following screen.
- Click [Next Step] to proceed.

Convert to Numerical Model

Now you are ready to populate the numerical grid/mesh with the conceptual elements.

- Click on the [Convert to Numerical Model] button to proceed
- This conversion could take several minutes, depending on the size and type of grid you used, and the complexity of the conceptual model inputs.
- A new window displaying the conversion progress will open. You should see a message indicating that the model conversion has completed. Click [Close] to dismiss this window.
A new workflow tab (Numerical Modeling) will open in your project with these steps:

- Keep the default modeling objectives and click [Next Step] to proceed to the Properties step.
View/Edit Properties

- At this step, you can view/edit the flow properties for the model.

- Under Views, select the various views you want to see in the Flex viewer; VMOD Flex allows you to simultaneously show a layer, row, column and 3D Views. Place a checkbox beside the desired view and it will appear on screen.

- Adjust a specific layer, row, or column using the up/down arrows. Alternatively, click on the button then click on any specific row, column, or layer in any of the 2D views, and the selected row, column, or layer will be set automatically.

- Now you will define a default initial heads value.

- Choose [Initial Heads] from the combo box under the Toolbox as shown below
Click [Edit...] button located below the Initial Heads combo box.

Type ‘350’ in the top ‘InitialHeads (m)’ cell.

Then press [F2] (or the [ ] button) to propagate this value to all other cells in this column; this will apply an initial head value of 350 for the entire model domain.

Click [OK] when you are finished.

Use the same tools as described in the previous step to manipulate the views.

The display tools located above the grid viewer window allow you to switch from discrete cells rendering to color shading/contours.

**Please Note:** this is available only when you do attribute rendering, and not when you are rendering by ZoneID.
In the Toolbox, you can select a different parameter group and see the corresponding zonation in the Flex Viewers. For example, try turning on the column view and switching to Conductivity to see the two zones you defined earlier.

- Click [Next Step] (Next Step) to proceed to the Define Boundary Conditions step.

**Define Boundary Conditions**

At this step, you can view/edit the flow boundaries for the model.

- From the Objects in view window, select the Desired Boundary condition group (Constant Head, Rivers, etc.).
- Then select [Edit...].
- Click on a cell that belongs to this group; a dialog will appear where you can see the values for the boundary you selected.
- Click [OK] to close the view.
- Click [Next Step] (Next Step) to proceed. You will arrive at the 'Selection' step.
Proceed to Run or Define Optional Model Elements

You will arrive at a choice screen; here you can proceed to some of the “non-essential” inputs for the model, such as Zone Budget Zones, Particle Tracking, or Observation Wells. Or, you can proceed to Running the simulation.

- Click the [Select Run Type] button to proceed (Mouse over this and you will see the blue “Next” arrow appear on top; just left click once to select this option. (Alternatively, the [Next step] button will take you to this step, as it is pre-defined as the default step.)
Click the [Single Run] button to proceed (Alternatively, the [Next step] button will take you to this step).

You will arrive at the 'Select Engines' step. Here you can choose what engines you want (what version of MODFLOW: 2000, 2005, etc..), and if you want to include MODPATH and ZoneBudget in the run.

- MODFLOW-2005 should be selected by default.
- Click [Next Step] (Next Step) to proceed.

**Translate Packages**

- You will arrive at the 'Translation Step'.
At this step, you choose if the model is steady-state or transient, choose the solver you want to use, and define any other MODFLOW package/run settings, such as cell-rewetting, etc.

**Please Note:** in the General Settings, there is a default location where the MODFLOW and other files will be generated.

- Click the [ ] button near the [ ] button to proceed; this will read the input from the numerical model and “translate” this into the various input files needed by MODFLOW and the other engines. The files will be created in the directory defined in the previous step.
Click the [Next step] button to proceed. You will arrive at the “Run Engines Step”.

Run Engines

Click the [Run] button near the [Next step] button on the main workflow toolbar to start running the engines. You will see the Engine progress in the scrolling window.
Please Note: after a successful run, the Heads and Pathlines items will be added the tree in the Model Explorer tree in the lower left of the application window.

- Once finished, Click the [Next step] button to proceed.

View Results

- You can then choose to view results in the form of Maps (Contours and Color shading) or Charts
- Click the [View Maps] button.
- Hit F4 to hide the Workflow tree and make more viewing area for the maps.

**Please Note:** you can turn the workflow tree back on by hitting F4 at any time.

- Make sure in views you only have Layer checked "on". By default, the maps always show heads first. You can change this by checking one of the other output options in the Model Explorer:
You can see color shading of the calculated heads, in layer view.

You can display heads along a row, and along a column, and in 3D, using the same tools as you used earlier (refer to View/Edit Properties section).

If your model is transient (this exercise does not apply), you can use the time controls above the Flex Viewer to change the output time; as you do this, all active viewers (layer, row, column, 3D), will refresh to show the heads for the new output time.

The next section will discuss how you can generate a new grid with a different size and resolution, and generate a numerical model using this grid.

**Evaluating Different Grids**

Often the initial grid size you defined is not adequate to provide the solution and stability you require from your model. In this section, we explain how you can generate multiple grids from the conceptual model and run the corresponding numerical models.

- At the top of the grid view you will see a list of active tabs:
Click on the first tab, which should be your Conceptual Model workflow to make this the active window, and it should now appear on your display.

Click [Select Grid Type] from the workflow tree.

Click [Define Finite Difference Grid] button and the Define Grid window will appear.

Define a new grid with the desired grid size and rotation. (try a grid with twice as many rows and columns; i.e. 200 rows and 200 columns)

Click [Next>>].

Specify the desired vertical discretization; you may wish to use a different vertical grid type, or refine any of the vertical layers.

Click [Finish] when you are done.

The new grid should now appear, and you will also see the grid appear as a new node in the Model Explorer tree.

Click the [Next step) button to proceed.

Now you are ready to populate the numerical grid/mesh with the conceptual elements. The 'Convert to Numerical Model' display should appear similar as below. Now, in the 'Select Grid' combo box, you will see there are 2 grids; by default, the grid you just created should be selected.

Click on the [Convert to Numerical Model] button to proceed.

After clicking on the conversion button, a new workflow window will appear which includes the steps for the numerical model for this new grid.

Please Note: the new tab is titled with the name of the new grid you provided and
In addition, this new model run will appear in the model tree. The model run has a grid and corresponding inputs; this can also be seen in the figure above.

When the conversion is complete, click [Next Step] to proceed to the Properties step.

Now, as explained previously, you can review the properties and boundary conditions, and translate and run this model.

Once the heads are generated, you can compare this to the results from previous grids.

It's also very easy to generate different grid types (such as unstructured V-grids, or quadtree grids (Q-grids) when you use the conceptual modeling workflow. To test these alternate grid types return to the conceptual modeling workflow. On the 'Select Grid Type' workflow step you can select either the 'Define Unstructured V-Grid' or 'Define Unstructured Q-grid' option (the steps below are for a Q-grid example).

At the top of the grid view you will see a list of active tabs:

- Click on the first tab, which should be your Conceptual Model workflow ('Conceptual Model 1') to make this the active window, and it should now appear on your display.
- Click [Select Grid Type] from the workflow tree.
- Click [Define Unstructured Q-grid] button and the 'Create Unstructured Q-Grid' window will appear.
Let’s perform a simple refinement around the boundary conditions within the model. Using the table at the top of this window we will visualize the constant head and river boundary condition objects, and refine the cells which contain these boundaries to a desired size.

- Activate the ‘Visible’ checkbox for the river and constant head (East and West) boundary conditions
- Type ‘10000’ in the ‘Min Area (m^2)’ field for all three boundary conditions
- Click the ‘Refine to Min’ button for all three boundary conditions
- The resulting Q-grid should look like the image below:
Click 'OK' in the 'Create Unstructured Q-Grid' window
Proceed to the 'Convert to MODFLOW-USG Model' workflow step
Click on the [Convert to Numerical Model] button to proceed.

After clicking on the conversion button, a new workflow window will appear which includes the steps for the numerical model for this new grid.
When the conversion is complete, click [Next Step] (Next Step) to proceed to the Properties step.

Now, as explained previously, you can review the properties and boundary conditions, and translate and run this model.

Once the heads are generated, you can compare this to the results from previous grids.

**Please Note:** it will be necessary to update the initial head values to something more realistic (i.e. 350m) in the Q-grid model run, since this grid type does not support the full range of initial head settings supported by finite difference models (i.e. it is not possible to easily set the initial heads equal to ground surface in the MODFLOW-USG translation settings).

When the model runs successfully you should see the following results (map of heads in layer 1) for the Q-grid realization of your model:
As you can see, the conceptual modeling workflow is ideal for generating multiple realizations of your model using different grid types, different levels of grid refinement, etc. This makes scenario analysis easier than ever!

*****This concludes the 'Conceptual Modeling' tutorial.*****

3.2 Importing VMOD/MODFLOW Models

The following example is a quick walk-through of the basics of importing an existing Visual MODFLOW Classic or MODFLOW data set.

Objectives

- Learn how to create a project and import an existing numerical model
- Become familiar with navigating the GUI and steps for numerical modeling
- Learn how to view and edit properties and boundary conditions, in a variety of views
- Translate the model inputs into MODFLOW packages and run the MODFLOW engines
• Understand the results by interpreting heads and drawdown in several views
• Check the quality of the model by comparing observed heads to calculated heads

Required Files

This tutorial is designed to allow you to select your own Visual MODFLOW or MODFLOW project, and follow through the steps. If you wish to use the model that is shown in the following example, it can be downloaded from our website.

Before You Start!

If you need to create, modify, or maintain a model that utilizes any of the following features, you must continue to use Visual MODFLOW Classic interface for this:

- Flow Engines (MODFLOW-96)
- Flow Packages (ETS1, MNW, STR)
- Transport Engines (MT3D99, PHT3D)
- MGO

Importing a Visual MODFLOW Classic Project

The release of Visual MODFLOW Flex v5.0 introduced a new method of importing Visual MODFLOW Classic projects. This new method simplifies the process to import an existing Visual MODFLOW Classic project. The new method combines the 'Create a new project' and 'import model files' processes into a single step. This section describes the new procedure for importing Visual MODFLOW Classic projects. The old/customary method of importing Visual MODFLOW Classic and other MODFLOW projects is covered in the next section (Create the Project and Import Model Files).

If you import your Visual MODFLOW Classic project using this new method, please follow the steps in this section, and then skip ahead to the ‘View/Edit Grid’ section. Please note that this method does not work for other MODFLOW projects, it is specific to the import of Visual MODFLOW Classic projects.

- Launch Visual MODFLOW Flex.
- Select [File] then [Import Project..].
- Browse to the location of your Visual MODFLOW Classic project and select the Visual MODFLOW Classic file (i.e. wellhead-capture-zone.VMF file).
  - This project file will be contained in the 'Public Documents\Visual MODFLOW Flex\Tutorials\VMOD Import' folder.
• Click [Open].
• The 'Import Visual Modflow Classic Project' window will open:

![Import Visual Modflow Classic Project dialog](image)

• Enter a project name for the Visual MODFLOW Flex project (by default it will enter the same project name as the Visual MODFLOW Classic project).
  - In this example the project has been named 'VMOD Import Tutorial'
• Select a data repository for the Visual MODFLOW Flex project, select whether to create a folder for the project, and provide a description as required.
• Click [Import]
• A dialogue window will load showing the progress of the model import:

![Model import progress](image)
The dialogue window should indicate that all model elements have been imported successfully. The following model components will be loaded: grid, model properties, boundary conditions, MODPATH/ZoneBudget inputs (i.e. zones, particles) and observation data.

- Click [Close]
- The new numerical model should be fully loaded, and the 'Define Modeling Objectives' workflow step will be active:

You can now continue with the regular numerical modeling workflow by proceeding to the section of this tutorial titled View/Edit Grid.

Creating the Project and Import Model Files

This section describes the procedure for importing MODFLOW project files (developed independently of the Visual MODFLOW Classic/Flex interface) into the Visual MODFLOW Flex interface. This procedure can also be used to import Visual MODFLOW Classic projects, but the procedure outlined in the previous section (for Visual MODFLOW Classic projects) is more streamlined.

- Launch Visual MODFLOW Flex.
- Select [File] then [New Project..]. The Create Project dialog will appear.
- **Type** in the project name 'Import Tutorial'.
- Click the \[\text{\textbullet}\] button, and navigate to a folder where you wish your projects to be saved, and click [OK].
- Define your coordinate system and datum (or just leave the local cartesian as defaults).
- For this project, the default units will be fine.

The Create Project dialog should now look like this (units may be different):

![Create Project dialog](image)

- Click [OK]. The workflow selection screen will appear.
- Select **[Numerical Modeling]** and the Numerical Modeling workflow will load.
- In this step, you define the objectives of your model and the default parameters.
For the Start Date, expand the date picker, and select **1/1/2000**

**Please Note:** the start date for the numerical model is only relevant should you decide to import pumping well or observation data in absolute times; MODFLOW always uses relative time, with a start time = 0 (days or seconds)

- Click [ ] (Next Step) to proceed.
- At this step, you can choose to create a new empty numerical grid, or import an existing project.
- Click the [Import Model] button.
- Navigate to the folder that contains your Visual MODFLOW or MODFLOW project. Or you can open the project located in: C:\Users\Public\Documents\Visual MODFLOW Flex\Tutorials\VMOD Import
- Select the file (.VMF [VMOD Classic] or .NAM [MODFLOW] file) and click [Open] to continue. The import will start and you will see the status in the progress window.
  - For VMOD Classic projects it is usually easier to follow the steps listed in the section above (i.e. simply click File > Import Project... from the main menu, and browse to and select a VMOD Classic project file (.VMF))
  - If the project was built in a different groundwater modeling program than the model name file (.NAM) must be selected instead
- During the import, there are a few things to observe:
- Flex runs a series of validation checks on the MODFLOW projects you import and will display warnings for any anomalies it finds. In the example project provided for this tutorial, Flex will warn you that the wellheads-capture-zone.mbu file is missing from the project you are importing.
- Hit [OK] to continue.
The status of each model element import is shown in the main window; any detected errors will be shown here.

After the import, you will see the 'Model Explorer' is populated in the bottom left corner of the screen; from here, you can show/hide different model inputs/outputs.

Please Note: You can add other data objects to the model, such as an image or other raw data (polyline/polygon shapefiles). These files need to first be imported, then they can be displayed at the one of the subsequent steps ('View Grid', 'Define Properties', 'Define Boundary Conditions') or in a 3D viewer by selecting [Window] then [New 3D Window], then checking on the box beside the appropriate data object.

Click [Next Step] to proceed, where you will arrive at the View/Edit Grid step.

View/Edit Grid

At this step, you can view the numerical grid in layer (plan) view, cross-sectional (along row or column), and 3D view.

There are numerous tools available to control and manipulate the grid views:
Under 'Views', select the various views you want to see in the Flex viewer; VMOD Flex allows you to simultaneously show a layer, row, column and 3D Views. Place a check box beside the desired view and it will appear on the screen.

Adjust a specific layer, row, or column using the up/down arrows. Alternatively, click on the button then click on any specific row, column, or layer in any of the 2D views, and the selected row, column, or layer will be set automatically.

The standard navigation tools allow you to zoom, pan, and in the case of 3D view, rotate.

Click (Next Step) to proceed to the Properties step.

View/Edit Properties

At this step, you can view/edit the flow properties for the model.
Under the Toolbox, use the combo box to select from the various Property Groups: Conductivity, Initial Heads, and Storativity.

For each parameter group, you can choose to render by Zones or by a selected attribute. Based on your selection, the color rendering in the views will change.

Click [Edit...] button to see the conductivity zones that exist in your model and the values in each individual cell.

Use the same tools as described in the previous step to manipulate the views.

The display tools will allow you to switch from discrete cells rendering to color shading/contours.

The display tools will allow you to switch from discrete cells rendering to color shading/contours (note, this is available only when you do attribute rendering, and not when you are rendering by ZoneID)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Show/hide grid lines" /></td>
<td><img src="image" alt="Show as cells" /></td>
</tr>
<tr>
<td><img src="image" alt="Show as Surface" /></td>
<td></td>
</tr>
</tbody>
</table>
At the bottom of the display, you will see in the status bar the position of your mouse cursor in the current view (XY) grid position (Layer, Row, Column), grid dimension (cell width, length, and thickness), and the Zone ID or attribute value for the selected cell.

Click [Next Step] (Next Step) to proceed to the Boundary Conditions step.

**View/Edit Boundary Conditions**

At this step, you can view/edit the flow boundaries for the model. By default, Constant Heads (if any) will be displayed in the model. In order to see other boundary conditions, you can set these to visible from the Model Explorer (check on the appropriate cell groups or boundary condition category). Also, when you choose another boundary condition from the Toolbox (e.g., River), then this set of boundary condition cells will become visible.

- From the toolbox, select the desired boundary condition group (Constant Head, Rivers, etc.)
- Then select [Edit...].
- Click on a cell that belongs to the boundary you are interested in; a dialog will appear where you can see the parameters for all cells in that boundary. If you want to switch to see values in a different boundary, you can select it in the grid view and the dialog will automatically update to show the new boundary.
• For Recharge or Evapotranspiration, you need to select this Boundary Condition type from the list under the Toolbox, and the Recharge/EVT zonation will appear. A [Database] button will be enabled allowing you to view the zonations.

• You can also assign new boundary conditions to the model using the [Assign] options; for more details, please see Define Boundary Conditions.

• Click [Next Step] (Next Step) to proceed. You will arrive at the “Select the Next Step” options screen.

**Proceed to Run or Define Optional Model Elements**

• You will arrive at a choice screen; here you can proceed to some of the “non-essential” inputs for the model, such as Zone Budget Zones, Particle Tracking, and Observation Wells. Or, you can proceed to Running the simulation.

• Click the [Select Run Type] button to proceed (Mouse over this and you will see the blue [ ] arrow appear on top; just left click once to select this option. (Alternatively, the [ ] (Next step) button will take you to this step, as it is pre-defined as the default step.)
• Click the [Single Run button] to proceed. (Alternatively, the [ ] (Next step) button will take you to this step, as it is pre-defined as the default step.)

• You will arrive at the 'Select Engines' step. Here you can choose what engines you want (what version of MODFLOW: 2000, 2005, etc..), and if you want to include MODPATH and ZoneBudget in the run.

• The version of MODFLOW that you used in the VMOD Classic model should be selected by default; if you wish to run MODPATH and ZoneBudget, be sure to select these engines as well.

⚠️ Please Note: ***De-select MT3DMS engine.*** since there are no transport inputs defined for this model.

• Click [ ] (Next Step) to proceed.
Translate Packages

- You will arrive at the 'Translation Step'.
- At this step, you choose if the model is steady-state or transient, choose the solver you want to use, and define any other MODFLOW package/run settings, such as cell-rewetting, etc. For more details, see MODFLOW Translation Settings.

⚠️ Please Note: in the General Settings, there is a default location where the MODFLOW and other files will be generated.

- Click the [Translate] button to proceed; this will read the input from the numerical model and “translate" this into the various input files needed by MODFLOW and the other engines. The files will be created in the directory defined in the previous step.
Quick Start Tutorials 112

© 2019 by Waterloo Hydrogeologic

• Click the [Next step] button to proceed. You will arrive at the “Run Engines Step”.

**Run Engines**

• Click the [Run] button on the main workflow toolbar to start running the engines. You will see the Engine progress in the scrolling window.
Note that after a successful run, the Heads and Pathlines items will be added the tree in the model explorer.

Once finished, Click the [Next step] button to proceed.

View Results

You can then choose to view results in the form of Maps (Contours and Color shading) or Charts.
View Maps

- Click the [View Maps] button.
- You will then see color shading of the calculated heads, in layer view.
• You can display heads along a row, and along a column, and in 3D, using the same tools as you used earlier. (refer to View/Edit Grid section)

• If your model is transient, use the time controls above the Flex Viewer to change the output time; as you do this, all active viewers (layer, row, column, 3D), will refresh to show the heads for the new output time.

• If you ran MODPATH, you will see Pathlines appear as a new node in the ‘Model Explorer’ tree under Output; add a check box beside the Backward Pathlines (i.e. [Points Group1 Pathlines]) to display these in the active 2D/3D Viewers. You may need to zoom into the middle of the layer view to see the pathlines.

Please Note: you may need to revise the particle settings before running MODPATH. Right-click the existing particle group object in the Model Explorer and select ‘Edit’ to change particle group settings. Remember that backward particles should have a release time at the end of the simulation and an end time at the beginning of the simulation.

The pathlines are well represented in a 3D Viewer.
- Turn on the check box beside 3D View, in the available views
- Turn off the check box beside Layer View
- From the Model Explorer, turn off the check box beside Heads
- From the Data explorer (raw data), turn on the check box beside "VMod Imported Wells"
- Take a moment to rotate your 3D view, and zoom in, and you can get a display similar to the one shown below.

You can also display velocity vectors, which will allow you to easily display and interpret flow fields.

- Turn off the check box beside 'Points Group 1 Pathlines' in the Model Explorer
- Turn on the 'Velocity' output option, in the Model Explorer under 'Outputs' > 'Flow'
- Access the Velocity display settings by right-clicking and selecting 'Settings'
- In the Settings window, access the 'Style' > 'VelocityMap' settings

The VelocityMap settings window should appear as below:
Velocity vectors may be displayed using average linear or Darcy velocities. In-plane velocity vectors can be displayed using the full magnitude of the vector (i.e. the out-of-plane velocity component will be included), as a projection (i.e. only the in-plane velocity component is displayed), or simply a directional indicator (i.e. the size of the arrow is not dependent on the actual velocity). By default, the velocity vectors will initially be displayed as a projection with average linear velocity values, on a linear scale.

A normal velocity color map function is also supported, which allows you to interpret flows perpendicular to the selected layer/row/column (i.e. out-of-plane flow). By default the normal velocity color map will be displayed with a red/blue color scheme, with red areas indicating flow inward (i.e. corresponding to the positive X, Y or Z direction), and blue areas indicating flow outward (i.e. corresponding to the negative X, Y or Z direction). Areas with velocities below the specified in-plane range threshold will be displayed using the specified in-plane color (which is grey by default). You can find more information about the velocity map display settings in the 3D Gridded Data section of this manual.

Update the velocity map settings as follows:
• In the 'Color' frame, select the 'Custom' option, and select 'Yellow' as the display color
• In the 'Velocity type' frame, select the 'Darcy' option
• In the 'Scale factor' frame, type a scale factor of 3.0
• Deselect the 'Show legend' option, so that the legend will be hidden
• Take a moment to rotate your 3D view, and zoom in, and you can get a display similar to the one shown below.

View Charts

• Click on View Charts from the workflow tree, and a blank charting window will appear.
• Select the appropriate set of wells, or layers that contain wells, and click [Apply] and the following window will appear:
To the left of the chart window, you can choose what observation data to view; select individual wells, or see wells that belong to a specific group. After making a change to the well(s) selection, click on the [Apply] button to update the chart.

In the Charts toolbox, under Chart Type, select [Time Series], and the following window will appear:
3.3 Airport Numerical Model with Transport

The following example walks through creating a numerical model with groundwater flow (using MODFLOW-2005) and basic contaminant transport (using MT3DMS). The exercise is based on the well-known Airport example from Visual MODFLOW Classic.

Objectives

- Learn how to create a project and create a numerical grid
- Become familiar with navigating the GUI and steps for numerical modeling
- Learn how to define new property zones and boundary conditions
- Define inputs for contaminant transport
- Translate the model inputs into MODFLOW and MT3DMS packages

*****This concludes the 'Importing Visual MODFLOW/MODFLOW Models' tutorial.*****
- Run MODFLOW-2005 and MT3DMS engines
- Understand the results by interpreting heads, drawdown, and concentrations in several views
- Check the quality of the model by comparing observed heads to calculated heads, and observed vs. calculated concentrations

⚠️ Please Note: if you are unable to locate some supporting files for the tutorial, you may download these from the website.

Create the Project

- Launch Visual MODFLOW Flex.
- Select [File] then [New Project..]. The Create Project dialog will appear.
- Type in project name 'Airport Tutorial'.
- Click the [ ] button, and navigate to a folder where you wish your projects to be saved, and click [OK].
- Define your coordinate system and datum (or just leave the Local Cartesian as defaults).

For this project, the default units will be fine. The Create Project dialog should now look as follows:
• Click **[OK]**. The workflow selection screen will appear.
Select [Numerical Modeling] and the Numerical Modeling workflow will load.

The first step is to Define Modeling Objectives.

**Define Modeling Objectives**

- In this step, you define the objectives of your model and the default parameters.
- The **Start Date** of the model corresponds to the beginning of the simulation time period.
- Select 1/1/2000 for the **Start Date**
- For this scenario, we will include contaminant transport in the model run. **Click the check box beside 'Transport Active'** (in the right hand side of the window, under Define Modeling Objectives.
- For 'Retardation Model' select 'Linear isotherm (equilibrium-controlled)'. For this tutorial you will not be simulating any decay or degradation of the contaminant, so the default 'Reaction' setting of 'No kinetic reactions' will be fine.
- Below the 'Retardation Model' and 'Reaction' settings are two tabs: 'Species Parameters' and 'Model Parameters'. By default, one species (chemical component) is defined for the transport run. For this example, we will leave the initial concentration ('*InitialConcentration (mg/L)*') as zero, but adjust Kd (the distribution coefficient)
- Type 1E-7 in [Kd 1/(mg/L)] column
You are now finished setting up the flow and transport objectives. Click [Next Step] to proceed.

The following 'Define Numerical Model' step will appear; at this step, you can import Visual MODFLOW Classic or MODFLOW data sets, or define a new empty grid. For this tutorial we will create a new grid.

Click on [Create Grid] to proceed
Create Grid

- At this step, you can specify the dimensions of the Model Domain, and define the number of rows, columns, and layers for the finite difference grid. **Type** the following into the Grid Size section,
  - **Columns**: 40
  - **Rows**: 40
  - **For Grid Extents**, enter 2000 for Xmax and 2000 for Ymax
  - **Under Define Vertical Grid**, **Type 3 for Number of Layers**

Define Layer Elevations

- In Visual MODFLOW Flex, you can define the elevations of the tops and bottom of the model layers. Or you can have varying layer elevations defined from Surface data objects. Surfaces could be from data objects you imported from Surfer (.GRD, ESRI .ASI, .DEM), or from Surfaces you have created through interpolating XYZ points. In this exercise, you will import 4 surfaces (from Surfer .GRD files), then use these to define the layer elevations.
  - Click [File] then [Import Data...] from the main menu. The following window will appear:
For the Data Type, select 'Surface' from the drop-down list.

In the Source File field click the [...] button and navigate to your Public "My Documents" folder, then "VMODFlex\Tutorials\Airport\SuppFiles\Surfer\airport-ground-surface.grd" and select [Open]

Click [Next>>]
Click [Next>>] (accept the defaults)
Click [Next>>] (accept the defaults)
Click [Finish]. You should now see a new "airport-ground-surface" data object appear in the data tree, in the top left corner of the window.

Now, repeat the above steps to import the other Surfer .GRD files into the project:
- airport-layer2-top.grd
- airport-layer3-top.grd
- airport-layer3-bottom.grd

When you are finished, you should see 4 Surface data objects in the data tree in the top left corner.
Now you are ready to define the grid layers using these surfaces. Under Number of Layers, select 'Use Surface' check box for each grid layer. This is shown below.

![Number of Layers Table](image)

Next you will provide a surface for each layer;

- Click on **airport-ground-surface** from the data object tree (it should become selected), then click on the topmost blue arrow [ ] beside 'Use Surface' in the Number of Layers table (in the row that starts "Layer 1 - Top"). If you have done this correctly, the table should appear as shown below.

![Number of Layers Table](image)

Now repeat these steps for the remaining layers:

- Select **airport-layer2-top** Surface data object to the tree, and insert this (using the [ ] ) as the Surface for Layer 2 - Top
- Select **airport-layer3-top** Surface data object to the tree, and insert this (using the [ ] ) as the Surface for Layer 3 - Top
- Select **airport-layer3-bottom** Surface data object to the tree, and insert this (using the [ ] ) as the Surface for Layer 3 - Bottom
- When you are finished, the table should appear as shown below.
You are finished defining the layer elevations.
Click on the [Create Grid] button (near the top right of the window) to create the grid. You will see the model tree will be generated on the left side of the window, and 'NumericGrid1' should appear as the last item.

Refine the Grid

This section describes the steps necessary to refine the model grid in areas of interest, such as around the water supply wells, refueling area, and area of discontinuous aquitard. The reason for refining the grid is to get more detailed simulation results in areas of interest, or in zones where you anticipate steep hydraulic gradients. For example, if drawdown is occurring around the well, the water table will have a smoother surface if you use a finer grid spacing. Also, layer properties can be assigned more correctly on a finer grid.

- Right click on the 'NumericalGrid1' from the tree, and select [Edit Numerical Grid...]
- The following window will appear.
The grid refinement works by defining a starting row number, and ending row number, then a 'Refine by' factor; to help you define the limits of where the refinement should be applied, you can add data objects to this display, such as well locations, aerial maps, shapefiles, etc. When you are using this feature with your own models, you will need to import these data object files before starting the 'Grid Refinement' step.

- You will first start by refining the rows.
- Enter '5' in the 'From' field, and enter '35' in the 'To' field.
- Enter 'Replace Every [1] row(s) with [2]'. Your screen should appear as follows:
Click on the [Apply grid edit] button.
Next, you will refine the columns.
At the top left of the window, select the [Edit Columns] radio button.
As before, enter '5' in the 'From' field, and enter '35' in the 'To' field.
Enter Replace every [1] column(s) with [2]
Click on the [Apply grid edit] button.

You should now see coarse grid sizes around the edge of the model domain, and a more finer sized grid spacing in the middle of model (around the areas interest). This is shown below. The band of pink cells around the edges of the refinement indicate cells where the Max ratio threshold quality indicator (which is set by default to a cell step size of 1.50) has been exceeded. This can sometimes result in larger computational times and the potential for increased grid based dispersion. These areas can be further fractionally refined to improve the grid quality; in this exercise, we will leave the grid as-is and proceed to the next step.
In the Model Explorer tree, a new grid and numeric workflow (i.e. 'NumericGrid1_refined') is created, and a new workflow window/tab (i.e. 'NumericGrid1_refined-Run1') opens. At this stage we will continue working with the refined grid, and we can ignore the initial coarse grid.

- Now is a good time to save the project. Click [File] then [Save Project] from the main menu.
- In the next section, you will view the numerical grid that you just created.
- Click on the Define Properties step or click the next button [ ] to proceed.

**Define Flow Properties**

This section will guide you through the steps necessary to design a model with layers of highly contrasting hydraulic conductivities.

- First, ensure that 'Conductivity' is selected in the first dropdown menu under the 'Toolbox'
- Click on the [Edit...] button and Type the following values in the top row of the window:
  - Kx (m/s): 2E-4, then use [F2] to propagate through all cells
  - Ky (m/s): 2E-4, then use [F2] to propagate through all cells
  - Kz (m/s): 2E-4, then use [F2] to propagate through all cells
- Click [OK] to accept these values.

⚠️ Please Note: it may take a moment for Flex to process this change.

In this case the Kx, Ky, and Kz values are the same, indicating the assigned property values are assumed horizontally and vertically isotropic. However, anisotropic property values can be assigned to a model by modifying the Conductivity Database.

In this three layer model, layer 1 represents the upper aquifer, and layer 3 represents the lower aquifer. Layer 2 represents the aquitard separating the upper and lower aquifers. For this example, we will use the previously assigned hydraulic conductivity values (Zone# 1) for model layers 1 and 3 (representing the aquifers) and assign different Conductivity values (i.e. a new Zone) for model layer 2 (representing the aquitard). Note that layer 1 is the top model layer.

Next you need to change to Layer 2. (using the up arrow under the Layer text box shown below)

You are now viewing the second model layer, representing the aquitard. The next step in this tutorial is to assign a lower hydraulic conductivity value to the aquitard (layer 2). We can graphically assign the property values to the model grid cells.

- Click [Assign] then [Entire Layer/Row/Column] from the toolbox.

The following dialog will appear:
Click on the [New] button at the top; this will create a new zone.

Enter the following values:
- **Kx (m/s): 1E-10**
- **Ky (m/s): 1E-10**
- **Kz (m/s): 1E-11**

The dialog should appear as shown below:

- Click [OK] to accept these values.
Once finished, the cells for Layer2 should change blue, which indicates these cells belong to Zone2; you can use the Legend under the toolbox as a guide, and also mouse over cells in the grid view, and note the values in the status bar.

Next you must assign the appropriate conductivity values to the discontinuous region. Although the region where the aquitard pinches out is very thin, the conductivity values of these grid cells should be set equal to the Conductivity values of either the upper or lower aquifers. In this particular example, the zone of discontinuous aquitard is indicated on a shapefile. We will import this shapefile into the project:

- Click [File] then [Import data...]
- For the Data Type, select Polygon from the drop-down list.
- In the Source File field click the [...] button and navigate to your Public "My Documents" folder, then 'VMODFlex\Tutorials\Airport\suppfiles\discontinuous-aquitard.shp', and click [Open]
- Click [Next>>]
- Click [Next>>] (accept the defaults)
- Click [Next>>] (accept the defaults)
- Click [Finish].
- You should now see a new data object, 'discontinuous-aquitard' appear in the data tree, in the top left corner of the window.
• Click on the box beside this data object in the tree.
• The data object should now appear in the Layer View of the grid.
• Zoom into this area (using the mouse wheel, or the Zoom in button on the toolbar).
• Click [Assign] then [Using Data Object] from the toolbox.
• Use the button to pull in the discontinuous-aquitard shape file in the new property zone window (you can also drag/drop a data file into this field).
• We want to assign these cells to an existing Zone: Zone1 that represents the aquifer above. So in this case, it is not necessary to create a new parameter zone. Use the zone arrows to select Zone 1 and verify that the K-values are all 0.0002 and that Layer 2 is selected. Your screen should look like the following:

![Schematic image](image)

• Click [OK] to assign this group of cells to Zone1. This display should appear as shown below.
Now view the model in cross-section to see the three hydrogeological units. First, zoom out to the fill extent using one of the following options:

- ![Zoom out button from the toolbar](image)
- ![Zoom full extents button from the toolbar](image)
- Scroll wheel on the mouse (scroll forwards)

- Click the [Column] check box in the ‘Views’ section. You will see another view appear beside the layer view, showing a cross-section through the model domain (by default, through Column1). To improve this view, you should change the Exaggeration.
- **Enter 40 in the Exaggeration field**, which is located in the toolbar directly above the Column view window.
- **Enter 37 for the Column number**, as this will provide a cross-section through the region with the discontinuous aquitard. Take a moment to view the cross-section of the properties. You can also change the cross-section view (change the Column number up, down, or enter a new value), and use the zoom and pan tools on the Column view to improve the display. Note that you can repeat the same steps above for Rows, instead of Columns, in order to see cross-sections along the X-axis.
Define Boundary Conditions

The next step is to define the flow boundaries for the model. In this example you will define constant heads along the north, in layer 1 and 3, and the Waterloo River along the southern edge in layer 1, and a constant head along the south in layer 3.

Constant Heads

The first Constant Head boundary condition will be for the upper unconfined aquifer along the northern boundary of the model domain. To do this you will use the [Assign]-[Polyline] tool.

- First, you need to go back to Layer 1. Type '1' into the [Layer] field under 'Views' (or use the arrow buttons).
Next, ensure that 'Constant Head' is selected from the first dropdown menu under the 'Toolbox' section.

Under the 'Toolbox' section, click [Assign] > [Polyline]. Move the mouse pointer to the north-west corner of the grid (top-left grid cell) and left-click on this location to anchor the starting point of the line. Now move the mouse pointer to the north-east corner of the grid (top-right grid cell) and Right-Click on this location to indicate the end point of the line. Click 'Finish', then you should then see a small menu appear to 'Define Boundary Condition'.

Click [Next >>] to accept the default name.

Enter Starting Head (m) of 19 in the top row and hit F2 to propagate this value.

Enter Ending Head (m) of 19 in the top row and hit F2 to propagate this value.

Leave the default value of -1 for Conc001; this indicates that no contaminant mass will be assigned to these cells.

Click [Finish] to complete the boundary condition. The hand-drawn polyline will now turn to a set of red points, indicating that a Constant Head boundary condition has been assigned to these cells. The display should look like the image below:
Next you will assign a Constant Head boundary condition along the northern boundary for the lower confined aquifer.

- Locate the [Layer] selection in the 'Views' section, and change this to '3'.

- Ensure that 'Constant Head' is still selected from the first menu under the 'Toolbox' section.
- Click [Assign]-[Polyline] from the toolbox. Move the mouse pointer to the north-west corner of the grid (top-left grid cell) and left-click on this location to anchor the starting point of the line. Now move the mouse pointer to the north-east corner of the grid (top-right grid cell) and right-click on this location to indicate the end point of
the line. Click 'Finish' then you should then see a small menu appear 'Define Boundary Condition'.

- Click [Next >>] to accept the default name.
- In the Define Boundary Condition dialog, enter the following values:
  - Enter Starting Head (m) of '18' in the top row and hit F2 to propagate this value.
  - Enter Ending Head (m) of '18' in the top row and hit F2 to propagate this value.
- Click [Finish] to complete the boundary condition. The hand-drawn polyline will now turn to a set of red points, indicating that a Constant Head boundary condition has been assigned to these cells.

Next, assign the Constant Head boundary condition to the lower confined aquifer along the southern boundary of the model domain.

- Click [Assign]>>[Polyline] from the toolbox. Move the mouse pointer to the south-west corner of the grid (bottom-left grid cell) and click on this location to anchor the starting point of the line. Now move the mouse pointer to the south-east corner of the grid (bottom-right grid cell) and right-click on this location to indicate the end point of the line. Click 'Finish' then you should then see a small menu appear 'Define Boundary Condition'.
- Click [Next >>] to accept the default name.
- In the Define Boundary Condition dialog, enter the following values:
  - Enter Starting Head (m) of '16.5' in the top row and hit F2 to propagate this value.
  - Enter Ending Head (m) of '16.5' in the top row and hit F2 to propagate this value.
- Click [Finish] to complete the boundary condition. The hand-drawn polyline will now turn to a set of red points, indicating that a Constant Head boundary condition has been assigned to these cells.

**River**

The following instructions describe how to assign a River boundary condition in the top layer of the model, along the southern edge of the model.

- First, you need to go back to Layer 1 (using the steps explained previously).
- Now we need to import our river data object (using the steps explained previously; [File]>>[Import Data]>>'Polyline'>>'river.shp')
  - First, you need to go back to Layer 1. Type '1' into the Layer field under 'Views' (or use the arrow buttons).
  - Select 'River' from the first dropdown menu in the 'Toolbox' section.
  - Click [Assign]>>[Using Data Object] from the toolbox. Use the [button to add the river data object to the geometry section of the 'Define Boundary Condition' dialog. Hit [OK] to the message that the extents will be clipped to the model domain.
- Click [Next>>] to accept the default name.
- The 'Define Boundary Condition' window will appear as shown below.
Traditionally, the River boundary condition has required a value for the Conductance of the riverbed. However, the Conductance value for each grid cell depends on the length and width of the river as it passes through each grid cell. Therefore, in a model such as this, with different sizes of grid cells, the Conductance value will change depending on the size of the grid cell. In order to accommodate this type of scenario, Visual MODFLOW allows you to enter the actual physical dimensions of the river at the Start point and End point of the line, and then calculates the appropriate Conductance value for each grid cell according to the standard formula.

Select the first row in the define Boundary Condition dialog and notice Flex has highlighted in yellow the corresponding river cell (lower left corner of the river). This dialog is interactive; you can select cells in the dialog or viewer/grid window to see how boundary values are currently assigned.

- In the top row of the boundary assignment dialog enter the following values for the start point.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>River Stage (m)</td>
<td>16.0</td>
</tr>
<tr>
<td>Riverbed Bottom (m)</td>
<td>15.5</td>
</tr>
<tr>
<td>Conductance (m²/2/day)</td>
<td>(do not enter any values)</td>
</tr>
<tr>
<td>River Width (m)</td>
<td>25</td>
</tr>
<tr>
<td>Riverbed Thickness (m)</td>
<td>0.1</td>
</tr>
<tr>
<td>Riverbed Kz (m/s)</td>
<td>1E-4</td>
</tr>
</tbody>
</table>

💡 Leakance from the river will be calculated based on the parameters you define. For more details on the calculation, refer to Boundary Conditions Theory

- Now you will define the values for the End Point of the river.
- Scroll to the bottom row in the Define Boundary Condition dialog and click in it. Notice the river end cell is now highlighted yellow in the grid view.
Now define the values at this end point, in the parameters grid, based on the values below:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>River Stage (m)</td>
<td>15.5</td>
</tr>
<tr>
<td>Riverbed Bottom (m)</td>
<td>15.0</td>
</tr>
<tr>
<td>Conductance (m²/d)</td>
<td>(do not enter any values)</td>
</tr>
<tr>
<td>River Width (m)</td>
<td>25</td>
</tr>
<tr>
<td>Riverbed Thickness (m)</td>
<td>0.1</td>
</tr>
<tr>
<td>Riverbed Conductivity (m/s)</td>
<td>1E-4</td>
</tr>
</tbody>
</table>

We will now linearly interpolate the intermediate values. To do this:

1. **Click the grey box next to the top (start) row** to highlight the whole row blue as shown below:

2. **Holding the CTRL key on your keyboard, click to also highlight the last (end) row.**
3. **Now click the interpolate button [ ]**. The intermediate rows should now populate with interpolated values.
4. **Click [Finish]** to complete the boundary condition. The polyline will now turn to a set of blue points, indicating that a River boundary condition has been assigned to these cells. The interface should now look like the image below:
Recharge

In most situations, aquifers are recharged by infiltrating surface water. In order to assign recharge in Visual MODFLOW Flex, you must be viewing the top layer of the model. Check the Navigator Cube in the lower left-hand side of the screen to see which layer you are currently in. The first boundary condition to assign is the recharge flux to the aquifer.

- First, you need to ensure you are viewing Layer 1. Type ‘1’ into the [Layer] field under ‘Views’ (or use the arrow buttons).
- Select ‘Recharge’ from the list of boundary conditions in the toolbox.
- Click [Assign]→[Entire Layer] from the toolbox. The ‘Define Boundary Condition’ dialog will appear.
- Click [Next >>] to accept the default name.
- Enter 100 in the ‘Recharge (mm/yr)’ column and hit F2 to propagate this value to all rows.
- Leave the default value of -1 for Conc001; this indicates that no contaminant mass will be assigned to the recharge flux.
- Enter 0.1 in the ‘Ponding (m)’ column and hit F2 to propagate this value to all rows, as shown below.
- Finally, change the Schedule from ‘Transient’ to ‘Static’
- The Define Boundary Condition window should now look like the image below:
Click [Finish]. All cells in the top layer will be assigned a recharge rate of 100 (mm/yr). The recharge boundary will be displayed as white dots.

Now you will assign a higher recharge value at the Refuelling Area where jet fuel has been spilled on a daily basis. First you need to import a polygon shapefile that delineates this area.

- Click [File] > [Import data...]
- For the Data Type, select 'Polygon' from the drop-down list.
- In the Source File field click the [...] button and navigate to your Public "My Documents" folder, then 'VMODFlex\Tutorials\Airport\suppfiles\refuelling-area.shp', and click [Open]
- Click [Next>>]
- Click [Next>>] (accept the defaults)
- Click [Next>>] (accept the defaults)
- Click [Finish]. You should now see a new data object, 'refuelling-area' appear in the data tree, in the top left corner of the window.
- Click on the box beside this data object in the tree. The data object should now appear in the Layer View of the grid (it is located in the top middle of the site).

**Please Note:** you may need to uncheck 'Recharge' from the Model Explorer tree to make the view less cluttered.

- Zoom into this area (using the mouse wheel, or the Zoom in button on the toolbar).
Click [Assign]->[Using Data Object...] from the toolbox. A 'Define Boundary Condition' window will appear. Use the button to add the 'refuelling-area' object to the geometry section of the dialog.

- Click [Next >>] to accept the default name.
- Enter 250 in the 'Recharge (mm/yr)' column and hit F2 to propagate this value to all rows.
- Enter 0.1 in the 'Ponding (m)' column and hit F2 to propagate this value to all rows.
- Finally, change the Schedule from 'Transient' to 'Static'.
- Notice that Conc001 has a default of -1 indicating that there is no defined mass flux assigned to this boundary condition. You will modify this later on in the Transport section of the tutorial.

- Click [Finish]. You should now see a new zone of cells colored blue, indicating the new RechargeZone2, with this new value.
- Click [Database] to see the recharge zones you created, and their corresponding values.
Click [OK] to close the window.
Now is a good time to save the project. Click [File] > [Save Project] from the main menu.

Define Pumping Wells

To generate a pumping well boundary condition, you can either add them one at a time through the user interface, or use a wells data object for this model, you will begin by importing a wells data object.

Click [File] > [Import Data...] from the main menu.
Select 'Well' as the data type.
[...] to choose the source file
Browse to your Public "My Documents" folder, then 'VMODFlex\Tutorials\Airport\suppfiles\Pumping_Wells.xls' file.
[Open]
[Next>>]. The next window will show a preview of the data to be imported.
VMOD Flex provides you with various options to import well data. In this window, you must select to import the well heads, screens, and pumping schedules.

Select the [Well heads with the following data] check box.
Select the [Pumping Schedule] check box.

[Next>>]

[Next>>] to accept the default Coordinate System.

The following 'Data Import' window will then appear:
In this screen, you need to map the fields from the spreadsheet to the required fields in the data import utility. If you prepare your Excel file with the exact column names that are expected by VMOD Flex, then no mapping is required and this can save you time. For this exercise, the source Excel file has the map names pre-defined. Take a moment to review the required fields for the Wells import:

- **Well heads**: Well ID, XY Coordinates, Elevation, and Bottom
- **Screens**: Screen Id, Screen top Z, Screen bottom Z
- **Pump Schedule**: Pumping start date, Pumping end date, Pumping rate

💡 When working with your own pumping well data for your models, you can use this Excel file as a template; by having all the fields automatically mapped this reduces the effort required during the import process, and minimizes the source of errors.

Switch to the Screens Tab to see the mapped fields.
• Click [Next>>].

The Data Import preview will appear:
You should see a series of green check marks next to the 'Heads', 'Screens' and 'Pump Schedule' tabs indicating that there were no import errors.

Click [Finish].

The 'Pumping_Wells' will now appear as a new data object in the Data tree.

Next, you need to add these wells to the Numerical Model:

- At the Define Boundary Conditions step in the workflow, under Toolbox, choose 'Wells' from the first dropdown menu listing available boundary condition types.
- Click the [Assign]>[Using Data Object] button. A 'Create Well Boundary Condition' window will appear.
- Select (highlight) the 'Pumping_Wells' data object from the Data Tree (you may need to move the Pumping Wells Boundary Condition dialog to the right in order to see this).
- Click [ ] button located in the middle of the 'Create Well Boundary Condition' window, under 'Select Raw Wells Data Object or Drag & Drop'. Once completed, your display should appear as shown below.
Click [OK]. The pumping wells have now been added to the numerical model.

You should see a new node appear on the Model Explorer, under 'NumericGrid1_refined/Run/Inputs/Boundary Conditions/Wells'. In order to see these wells, you need to turn off the Recharge coverage and change to layer 3.

- Click on the box beside 'Recharge' in the Model Explorer, to remove the check box.
- Change to Layer 3 (as explained earlier).
- Also, you may need to zoom out to the full grid extents, by selecting the [](Zoom to Full Extents) button on the toolbar above the grid.
- You should see the two points representing the wells, located in the lower right corner of the model domain, as shown in the following figure.
Now is a good time to save the project. Click [File]-[Save Project] from the main menu.

Click [Next Step] to proceed.
Select [Define Observation Wells]

Define Head Observations

Field observations of groundwater heads and fluxes are essential in order to calibrate the results obtained by MODFLOW. In this exercise, you will add several head observations wells, and analyze these against the corresponding calculated values after the model run is complete. The head observations are grouped into two files, one for layer 1 (observation wells 1-5) and another for layer 3 (observation wells 6-10). First we need to import the observation wells:

Click [File]-[Import Data] from the main menu bar
Ensure 'Well' is selected as the Data Type
[...] to choose the source file.
Browse to your Public 'My Documents' folder, then 'VMODFlex\Tutorials\Airport\suppfiles\Head_Observations_Layer1.xls' file.
[Open]
[Next>>]
A preview window will appear displaying the source data.

- [Next>>]. VMOD Flex provides you with various options to import well data.
- Choose the radio button [Well heads with the following data]
- Then select [Observations points]
- Then select [Observed heads]
- Ensure you have the options selected as shown below.
In this screen, you need to map the fields from the spreadsheet to required fields in the Wells Import utility.

- [Next>>]
- [Next>>] to accept the default Coordinate System

In this screen, you need to map the fields from the spreadsheet to required fields in the Wells Import utility.
To save time, you can prepare your Excel file with the exact filenames that are required by VMOD Flex, and then no mapping is required. For this exercise, the source Excel file has the map names pre-defined. Take a moment to review the required fields for the Wells import:

- **Well heads**: Well ID, XY Coordinates, Elevation, and Bottom
- **Observation points**: Logger Id, Logger Z, Head observation date, Observed head

The Data Import preview will appear, indicating if there were any errors with the file import. This file should import with no errors.

- **[Finish]**

The 'Heads_Observations_Layer1' will now appear as a new data object in the Data Tree. Take a moment and visualize this in the 3D Viewer.

Next you need to add these raw observation wells as observation points for the numerical model.

- Be sure that the 'Head_Observations' data object is selected in the Data Tree.
- Click on the ![button](button.png) button located under the toolbox in the 'Define Observation Wells' workflow step.
- The observation wells will be added to the display and the numerical model tree. You should see several green points in the model domain that represent the locations where...
head measurements were taken. (in this example, all the head observations are in layer 1).

Now you will repeat these steps to import the observation wells for layer 3 of the model. Follow the same steps above, but this time import the 'Head_Observations_Layer3.xls' file. When you import a second observation group you will be prompted to save the new wells to an existing well group, or to 'Create a new well group', as indicated in the image below. When you see this window, select the 'Create a new well group' option:
• Click 'Create a new well group'
• OK

Once the observations for layer 3 have been imported you can view them in layer 3, as shown below:

• Now is a good time to save the project. Click [File]->[Save Project] from the main menu.
• Click [Next Step] (Next Step) to proceed.

Select Run Type and Engines

• In the ‘Select Run Type’ step, select [Single Run]
• From the ‘Single Run’ step in the workflow, you will see 'USGS MODFLOW 2005 from WH' is selected along with 'MT3DMS' as the transport engine.
• For the first run, we will run the flow solution only, without transport.
• De-select the checkbox for 'Run Transport Engine'
• Click [Next Step] (Next Step) to proceed.
MODFLOW-2005 Translation Settings

At the Translate step you have the option to adjust the various parameters and flags for the MODFLOW packages and run time settings. Available options include: 'Settings' (General), 'Settings' (MODFLOW-2005), 'Solvers', 'Recharge and Evapotranspiration', 'Lake', 'Layers', 'Rewetting', 'Initial Heads', 'Anisotropy', 'Output Control' and 'Advanced Settings'.

- Select [MODFLOW-2005]->[Settings]
- Enter 7300 for 'Steady-State Simulation Time' (in the grid in the main window).

- Click [Translate] to create the MODFLOW-2005 packages. Check the log to confirm the translation has finished.
Run MODFLOW-2005

- Click the [ ] button to run MODFLOW-2005.

The model run should complete in a few seconds. Once finished, you should see "The run was successful" in the engine progress window. In addition, you will see several items will be added to the model tree under 'Output'. See the image below as an example:
• Now is a good time to save the project. Click [File]→[Save Project] from the main menu.

• Click [ ] (Next Step) to proceed.

View Maps (Heads and Velocities)

The following 'View Results' window will then appear; you have the option to View Maps or View Charts. We will start by viewing maps of heads.
Click [View Maps] button to proceed. You will then see color shading of the calculated heads, in layer view. You can press F4 to hide the workflow, and bring it back again at any time by pressing F4 again.
• If you do not like the default contour interval or line color, you can customize the contour map settings.
• To access the contouring options for Heads, right-click on 'Heads' from the model tree, and select [Settings...]. The following dialog will appear

![Contour Settings Dialog]

• From the Settings tree on the left, select 'Style' followed by 'Contour Lines'. This will expand the settings window and give you access to the Contour Line settings.
• Under 'Automatic contour', enter 0.5 for the contour interval, and start/finish the contour at 12/20. This is shown below:
• When you are finished, click [OK].
• This will apply the changes to the head contours in your current view.

💡 Preview Display Settings before Committing

All of the Settings windows have an [Apply] button in the lower right corner. This means you can apply the adjusted changes and see the impact in the current 2D or 3D view before you close the window. This makes it easier to obtain the desired display without having to open and close this window several consecutive times.

• You can also display heads along a row, along a column, or in 3D using the same tools that you used earlier.
In the 'Views' section activate the 3D view and deactivate the layer view. Apply an exaggeration of 40 and reorient your view so that you can see head values along a layer, column and row. You should see something similar to the image below:

- Take a moment to view the heads in other layers/rows/columns.

It is also possible to visualize the results of your flow model as a distribution of velocity vectors, which allows you to easily display and interpret flow fields. To display a velocity map do the following:

- First, ensure you are viewing 'Layer 3'. This layer contains the pumping well, and should contain a significant flow field in that region.
- Turn off the check box beside 'Heads' in the Model Explorer
- Turn on the 'Velocity' output option, in the Model Explorer under 'Outputs' > 'Flow'
- Access the Velocity display settings by right-clicking and selecting 'Settings'
- In the Settings window, access the 'Style' > 'VelocityMap' settings

The VelocityMap settings window should appear as below:
Velocity vectors may be displayed using average linear or Darcy velocities. In-plane velocity vectors can be displayed using the full magnitude of the vector (i.e. the out-of-plane velocity component will be included), as a projection (i.e. only the in-plane velocity component is displayed), or simply a directional indicator (i.e. the size of the arrow is not dependent on the actual velocity). By default, the velocity vectors will initially be displayed as a projection with average linear velocity values, on a linear scale.

A normal velocity color map function is also supported, which allows you to interpret flows perpendicular to the selected layer/row/column (i.e. out-of-plane flow). By default the normal velocity color map will be displayed with a red/blue color scheme, with red areas indicating flow inward (i.e. corresponding to the positive X, Y or Z direction), and blue areas indicating flow outward (i.e. corresponding to the negative X, Y or Z direction). Areas with velocities below the specified in-plane range threshold will be displayed using the specified In-plane color (which is grey by default). You can find more information about the velocity map display settings in the 3D Gridded Data section of this manual.

Update the velocity map settings as follows:
In the 'Color' frame, select the 'Custom' option, and select 'Orange' as the display color.

In the 'Velocity type' frame, select the 'Average Linear' option.

In the 'Scale factor' frame, type a scale factor of 3.0.

Deselect the 'Show legend' option, so that the legend will be hidden.

Once these settings have been applied, the Layer view should look like the image below (Layer 3).

You should be able to see larger velocity vectors in the bottom-right corner of the model, in the vicinity of the pumping wells. These indicate that groundwater flows are significantly faster in this area. You may also notice a 'ring' approximately in the middle of the model. This indicates that there is significant flow perpendicular to layer 3 in this zone. Since the ring is blue, that would indicate flow in the negative Z direction, or down through layer 2 and into layer 3.

View Charts (Heads)

It's very easy to review calibration charts in Visual MODFLOW Flex whenever observation data has been applied to your model. Since we have applied observed head values in a previous step we can now proceed to the 'View Charts' workflow step to view the automatically generated calibration charts.
Click on ‘View Charts’ from the workflow tree, and the chart window will appear.
In the side toolbar, turn on the check box beside 'All Times' and 'All Obs'.
Click [Apply] and the chart should appear as shown below.

As you can see from the image above, Visual MODFLOW Flex supports a number of methods for selecting different wells. It will always be possible to turn on/off wells in particular layers. Visual MODFLOW Flex also supports observation 'groups'. In this example, our two observation groups have been delineated based on the layer in which they reside. However, you can group observation wells any way you like.

- Take a moment to view the correlation between calculated and observed heads
- Now is a good time to save the project. Click [File]–[Save Project] from the main menu.

In the next section of the tutorial, you will define the inputs for the transport run (properties and boundary conditions), then run MT3MDS along with MODFLOW-2005, and view and interpret the results.
Define Transport Model

The following section outlines the steps necessary to complete a simplified transport model. Similar to a groundwater flow model, a contaminant transport model requires properties (including initial concentrations), boundary conditions (sinks/sources), and observations in order to calibrate the transport model run against observed field conditions. These steps are described in the following sections.

Define Transport Properties

**Sorption**

In this example, the only reaction that will be simulated is linear equilibrium adsorption of a single dissolved species (the Jet fuel compound JP-4). However, depending on the complexity of the problem you are dealing with, your model may have several different zones with different sorptive and reactive properties (distribution coefficients, decay coefficients and yield coefficients) for each dissolved species in the model. In Visual MODFLOW Flex, the properties and processes for the transport model are assigned using the same types of graphical tools as you used for assigning the flow model properties.

For this tutorial you will not need to modify the Distribution Coefficient value you defined during the transport model setup, but you may examine the sorption parameter values as follows.

- From the workflow tree, click on [Define Properties] to go back to this step.
- Under the toolbox, choose [Species Parameter Conc001] from the first menu.
- Click on the [Zone] button and change it to [Kd]. The Parameters Database window will appear as shown below.

The default distribution coefficient (Kd = 1.0e-7 L/mg) was specified during the setup of the transport numeric engine. If this is not so (e.g. if you did not enter this value when creating the project), enter this now using the [Edit] button.

- With 'Species Parameter Conc001' and 'Kd' selected in the toolbox, click the [Edit] button.
- This will open the 'Edit property' window.
- Under the 'Kd' column enter 1E-07 in the first row and click F2 to propagate this value to the rest of the column.

The Kd values for each zone can be modified to accommodate heterogeneous soil properties and reactions throughout the model domain. However, for this example you will keep it simple and use a uniform Distribution Coefficient for each layer of the model.

**Dispersion Coefficients**

The next step is to define the dispersion properties for the model.
Visual MODFLOW Flex automatically assigns a set of default values for each of the dispersivity variables. The following table summarizes these values.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Dispersivity</td>
<td>10</td>
</tr>
<tr>
<td>Horizontal to Longitudinal Ratio</td>
<td>0.1</td>
</tr>
<tr>
<td>Vertical to Longitudinal Ratio</td>
<td>0.01</td>
</tr>
<tr>
<td>Molecular Diffusion Coefficient</td>
<td>0.0</td>
</tr>
</tbody>
</table>

It is possible to assign alternate values for the longitudinal dispersivity by using the [Assign] option buttons from the toolbox.

However, for this example, you will use a uniform dispersion value for the entire model domain.

In order to modify the horizontal or vertical dispersivity ratios and/or the molecular diffusion values you need to load the Layer Options.

- Right-click on 'Dispersion' from the Model Explorer, under Inputs/Properties/Transport
- Select 'Dispersion Parameters'. The following window will appear:

![Dispersion Parameters Window](image)

These parameters can be modified on a "per-layer" basis. For this example you will not need to modify the defaults.

- Click [Cancel] to close this window
- Now is a good time to save the project. Click [File] > [Save Project] from the main menu.
- Click [Next Step] (Next Step) to proceed

**Define Transport Boundaries (Sinks/Sources)**

In this section you will define the location and concentration of the contaminant source. The source of contamination will be designated at the refueling area as a Recharge Concentration that serves as a source of contamination to infiltrating precipitation.
Transport Boundary Conditions: VMOD Flex vs. VMOD Classic

If you are used to working with Visual MODFLOW Classic, you will notice a difference in how transport boundary conditions are handled in Visual MODFLOW Flex. In VMOD Classic, transport boundary conditions were defined separately from the flow boundary conditions using the types Constant Concentration, Recharge Concentration, Evapotranspiration Concentration, and Point Source. In Visual MODFLOW Flex, the sink/source parameters for transport models (which are time and species concentrations) are defined as part of flow boundary conditions, which is a more natural representation. This means you do not define separate cell geometries for transport boundaries, you simply define species concentrations while defining the flow boundary conditions, where required. Constant Concentration is an exception to this rule, since it does not need to coincide with a prescribed flux, you will still see a “Constant Concentration” boundary condition type, allowing you to define the geometry (cells) and parameters (time and species concentrations) for this Boundary Condition type.

When Transport is active in your model run, and you define a new boundary condition, you will see parameters for Species Concentration as part of the Boundary Condition attributes (e.g., Conc001, Conc002, etc...). These will have a default value of -1, indicating that no mass sink/source is defined for this group of boundary condition cells. As soon as you change this value to 0 or greater, then these cells will be treated as sinks/sources.

Assign Species Concentrations for the Recharge Boundary

When you defined the flow model, you created a separate recharge zone that covers the refuelling area. Now you will add a defined species concentration to this recharge flux.

- Ensure you are viewing the 'Define Boundary Condition' workflow step.
- Check to ensure that you are viewing Layer 1.
- Select ‘Recharge’ from the list of available boundary conditions under the toolbox.
- You will recall there were two recharge zones created for the flow model: background recharge of 100 mm/yr covering the entire model top, and a small area over the refueling area with a higher recharge rate of 250 mm/yr. The mass of contaminants will be assigned only to this smaller recharge zone.
- If not already visible, you must make the Recharge zones visible. In the Model Explorer, locate the 'Recharge' node, under 'Inputs'> 'Boundary Conditions', as shown below.
- Click on the box beside ‘Recharge’ in the tree.
The recharge cells should now appear in the layer view of the grid.
Under the 'Toolbox' section, click the [Database] button and the following window will appear.
When the recharge zones were previously created, the values for the chemical species (Conc001) were left as undefined, indicated by -1. You will modify this for the smaller recharge area.

- Locate **Zone2** (the second row in the table)
- Enter **5000** for **Conc001** (thereby replacing the -1 value).
- Click [OK] to close the Database window
- Now is a good time to save the project. Click [File] >> [Save Project] from the main menu.
- Click [Next] (Next Step) to proceed
- Select [Define Observation Wells]

**Define Concentration Observations**

The final step before running the transport simulation is to add the three observation wells to the model to monitor the jet fuel concentrations at selected locations down-gradient of the refueling area. The first observation well (OW1) was installed immediately down-gradient of the refuelling area shortly after the refuelling operation started. The other two observation wells (OW2 and OW3) were installed two years later when elevated JP-4 concentrations were observed at the first well (OW1).

You will import the concentration observations from an Excel file.

- Click [File] >> [Import Data] from the main menu bar
- Ensure 'Well' is selected as the Data Type
- [...] to choose the Source File.
- Browse to your Public 'My Documents' folder, then 'VMODFlex\Tutorials\Airport\suppfiles\Concentration_Observations.xls' file.
- [Open]
- [Next>>]

A preview window will appear displaying the source data.
VMOD Flex provides you with various options to import well data. Choose the radio button [Well heads with the following data] radio button. Then select [Observations points] and then select [Observed concentrations].
In this screen, you need to map the fields from the spreadsheet to required fields in the Dells Import utility. To save time, you can prepare your Excel file with the exact filenames that are required by VMOD Flex, and then no mapping is required. For this exercise, the source Excel file has the map names pre-defined. Take a moment to review the required fields for the Wells import:

- **Well Heads**: Well ID, X/Y Coordinates, Elevation, and Well Bottom
- **Observation Points**: Logger ID, Logger Z, Concentration observation date, Observed concentration

[Next>>] to accept the default Coordinate System.

[Next>>]
The Data Import preview will appear:
The 'Concentration_Observations' will now appear as a new data object in the Data tree, as shown below.

Now you need to add these raw observation wells as observation points for the numerical model.

- Ensure that you are on the 'Define Observation Wells' step in the numerical modeling workflow.
- Be sure that the 'Concentration_Observations' data object is selected in the tree
• Click the button located under the toolbox in the 'Define Observation Wells' workflow step.
• A window will appear allowing you to select from available well groups, or to create a new well group:
• Select 'Create a new well group' and retain the default name

![Select Well Group]

• The observation wells ('Concentration Observations') will be added to the display and the numerical model tree, under NumericGrid1_refined/Run/Inputs/Observations
• Locate 'Concentration Observations', and click on the box beside this data object in the Model Explorer.
You should see several orange points in the model domain that represent the locations where head measurements were taken.
• Now is a good time to save the project. Click [File]-[Save Project] from the main menu.

• Click [ ] (Next Step) to proceed.

• Select [Single Run], and click [ ] (Next Step) to proceed

• At the ‘Single Run’ step, be sure to include **MT3DMS** in the engine run; place a check box beside this engine in the list.

• Click [ ] (Next Step) to proceed.

**MT3DMS Translation Settings**

This section will guide you through the selection of the advection method, solver settings, and output times that you will use to obtain the solution and results for the contaminant transport model.
Solution Method

- Expand the 'MT3DMS' item under the Translation settings, and select [Solution Method]. A Solution Method settings window will appear.

For this model run you will be using the Upstream Finite Difference solution method with the Implicit GCG Solver. The Upstream Finite Difference method provides a stable solution to the contaminant transport model in a relatively short period of time. The GCG solver uses an implicit approach to solving the finite difference equations, and is usually much faster than the explicit solution method.
• Click the button in the 'Advection Method' option and select [Upstream Finite Difference (UFD)]
• Click the button in the 'Use Implicit GCG Solver' option and select [Yes].
• The Implicit GCG Solver Settings window will appear in the lower half of the translation settings window, as shown in the image below:

![Image showing Implicit GCG Solver Settings window]

• Though the Upstream Finite Difference method and the Implicit GCG Solver are computationally efficient, the tutorial simulation tracks contaminant transport over a 20 year period. In order to speed up the modeling process, you will use a nonlinear time step. Type the following information in the fields at the bottom of the window.
  • Multiplier = 1.1
  • Maximum transport step size = 200, as in the image above

**Output Settings**

Next, you will define the output times at which you would like to see the contaminant transport modeling results.
• Under the 'MT3DMS' section in the Translation settings, select [Output Control]. The Translation settings will update as shown below.
• Enter **7300** for the 'Simulation time length'
• Enter **5000** for 'Max number of transport steps'
• The remaining defaults can be left as they were found, as shown in the image below:

![MT3D Output times](image)

**Output Times**

• For this tutorial you will define specified times at which you would like to see the transport simulation results.
• The bottom half of the [Output Control] frame can be used to specify output times
• Click the ![Add Row](image) button; repeat this 7 more times.
• Enter the following output times in the grid.

<table>
<thead>
<tr>
<th>MT3D Output times</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
</tr>
<tr>
<td>730</td>
</tr>
<tr>
<td>1460</td>
</tr>
<tr>
<td>2190</td>
</tr>
<tr>
<td>2920</td>
</tr>
<tr>
<td>3650</td>
</tr>
<tr>
<td>5475</td>
</tr>
<tr>
<td>7300</td>
</tr>
</tbody>
</table>

• Now is a good time to save the project. Click [File] > [Save Project] from the main menu.
• You are now ready to translate the inputs into the MT3DMS packages.
• Click ![Translate](image) to create the MODFLOW-2005 and MT3DMS packages. (this should take approximately 10-20 seconds)
• Click ![Next Step](image) (Next Step) to proceed.
Run MODFLOW-2005 and MT3DMS

- Click the [Run] button to run MODFLOW-2005 and MT3DMS.
- The MODFLOW model run should complete in a few seconds; the MT3DMS run should also complete in 5-10 seconds.
- Once finished, you should see "***** The run was successful. *****" in the engine progress window.
- In addition, you will see several items will be added to the model tree under 'Outputs'. You should also see 'Concentrations' added to the model output tree under 'Output/Transport'.
- Now is a good time to save the project. Click [File]>[Save Project] from the main menu.
- Click [Next Step] (Next Step) to proceed.
- Click [View Maps].

View Maps (Concentrations)

- By default, the Heads will be shown in the Maps view. In order to see the 'Concentrations', you need to turn off 'Heads' from the model explorer and turn on 'Concentrations'.
- Locate the 'Output' node on the model tree.
- Remove the Checkbox beside 'Heads'
- Add a checkbox beside 'Concentrations'

- In the viewer toolbar, activate the 'Cells' rendering option by clicking the [Cells] button
• The concentration contours will be plotted for the first transport output time (in this case the first transport output time is 1 day).

• In order to see the concentration results at the other output times, you need to advance the output time. Click on the time step buttons located on the toolbar above the 'Layer View', as shown below. Alternately, you can expand the list of output times, and navigate directly to the desired output time.

• This display will then update with a plot of concentration contours for selected output time

• Open the first output time, 1 day, and the concentrations in the first layer of the model should look similar to the following figure:
You can determine the risk that the contaminant front poses to the discontinuous aquitard by doing the following:

- Locate the data object "discontinuous-aquitard" from the tree, and turn it on. It should appear in the layer view. Take a moment to navigate through the other layers, to see the calculated concentrations.
- Move your mouse cursor to specific areas of the interest (such as in the discontinuous aquitard region), and note in the status bar the calculated concentrations for the selected cell.
- After 7300 days (20 years) of simulation time it is clear that the plume has migrated to the ‘hole’ in the aquitard.
- To see how the plume looks in cross-section, turn on the [Column] view, and enter column 25
- Advance the times to see the plume migrating the upper layers down to the lower layers.

**View Charts (Concentrations)**

In this section you will learn how to compare the observed concentration data to the concentration values calculated by the model.

- Select [View Charts] item from the workflow tree.
From the 'Parameter' combo box to the left of the main chart window, choose 'Transport'.
Then, under 'Chart Type' select 'Time Series'.
Select 'All Times' and 'All Obs.' from the 'Observations' frame on the left side of the window.
Click [Apply].
You should now be viewing the breakthrough curves for each of the three concentration observation wells defined earlier in the model (see following figure).

This time-series graph shows the calculated result using a colored line data series while the observation data is displayed only as data point symbols.

Now is a good time to save the project. Click [File] -> [Save Project] from the main menu.

*****This concludes the 'Airport Numerical Model with Transport' tutorial.*****

3.4 SEAWAT Tutorial

The following example walks through creating a simple numerical model with variable-density groundwater flow (using SEAWAT). The exercise is based on the well-known Elder model.
Objectives

- Learn how to create a project and create a numerical grid for SEAWAT simulations
- Learn how to define new property zones and boundary conditions
- Define inputs/boundary conditions for SEAWAT contaminant transport
- Translate the model inputs into SEAWAT input packages
- Run SEAWAT engines
- Understand the results by interpreting heads, drawdown, and concentrations in several views
- Check the quality of the model by comparing observed heads to calculated heads, and observed vs. calculated concentrations

References


Introduction

Saltwater intrusion is a common occurrence along coastlines throughout the world. Saltwater is denser than freshwater, and consequently will tend to migrate inland underlying freshwater zones when groundwater resources are depleted or in areas where the water table is low. A static equilibrium is reached when the freshwater discharge force balances the buoyancy force due to density. Visual MODFLOW Flex includes the SEAWAT Engine to simulate saltwater intrusion.

SEAWAT is a computer program for simulating three-dimensional, variable-density, transient ground-water flow in porous media. SEAWAT was designed by combining a modified version of MODFLOW-2000 and MT3DMS into a single computer program. SEAWAT contains all of the processes distributed with MODFLOW-2000 and also includes the Variable-Density Flow Process (as an alternative to the constant-density Ground-Water Flow Process) and the Integrated MT3DMS Transport Process.

This tutorial illustrates a common application for SEAWAT and is based on the Elder Problem published in the User’s Guide to SEAWAT (Guo and Langevin, 2002). The basic design of this SEAWAT exercise is shown in the figure below.
The Elder model is commonly used as a benchmark problem for variable-density groundwater flow, where fluid density is a function of salt concentration. This problem simulates salt-water intrusion and proliferation through a freshwater aquifer. Molecular diffusion is the sole mechanism of hydrodynamic dispersion during the simulation, which runs for 20 years. Salt from the constant-concentration boundary at the top of the model diffuses into the model domain and initiates a series of complex vortices that redistribute the salt mass throughout the model. A constant-concentration boundary with a value of zero is specified for the lowest layer in the model. Two outlet cells with constant head values of zero are specified for the upper left and right boundaries. These constant head cells allow salt to diffuse into the model by providing an outlet for fluid and salt mass (Guo and Langevin, 2002).

**Create the Project**

- Launch Visual MODFLOW Flex.
- Select [File] then [New Project..]. The Create Project dialog will appear.
- Type in project name ‘SEAWAT_Elder’.
- Click the [ ] button, and navigate to a folder where you wish your projects to be saved, and click [OK].
- Define your coordinate system and datum (or just leave the Local Cartesian as defaults).
- Most of the default units will be fine, but we will need to update the conductivity units.
- Select [m/\text{d}] as the Conductivity units
- Click [OK] button in the lower right corner of this window.

**Note:** It is recommended to select the folder “C:\Users\User_Name\Documents\VMODFlex” as the default location for your projects. You may then add a new folder to each individual modeling.
project. For the purposes of this project you may create the sub-folder SEAWAT_Elder. A new folder can be created automatically by selecting the ‘Create a folder for the project’ button under Data Repository.

Before clicking OK, the Create Project dialog should look like the image below:

![Create Project Dialog](image)

The following window will then appear:
The Select Modeling Scenario allows you to choose whether to proceed with the Conceptual or Numerical modeling workflow. The conceptual modeling workflow allows you to import data objects into Visual MODFLOW Flex and to build a conceptual site model (CSM) that is grid independent. The CSM can then be used as a starting point for several different numerical models. In other words, numerical models (i.e. with different grid types, engines, etc.) can be quickly and easily created based on the same conceptual model. This makes it easy for you to manage several different numerical models with slight variations.

Conceptual modeling is not covered in this exercise, so we will proceed with the numerical modeling workflow

- Select [Numerical Modeling] and the Numerical Modeling workflow will load.

Proceeding with the numerical modeling workflow will bring you to the first step in that workflow, which is to define your model objectives. This step allows you to specify whether you will be running a fully saturated or variably saturated model, whether it will be a constant or variable density model, whether contaminant transport will be included, which flow/transport engines will be utilized, etc.

The selection of these modeling objectives will determine which model- and species-specific transport properties must be defined (e.g. molecular diffusion coefficients, distribution coefficients, decay constants, etc.). It is also possible to define default property values at this time.
For the default project properties please input the values as shown below.

- **Select Flow type**: Saturated (Variable Density)
- **Select Simulation type**: Groundwater flow
- **Available Flow Engines**: USGS SEAWAT from WH
- **Select Start Date**: today
- **Type**: $K_x (m/d) = 0.411$ in the Default Project Property Settings frame
- **Type**: $K_y (m/d) = 0.411$ in the Default Project Property Settings frame
- **Type**: $K_z (m/d) = 0.411$ in the Default Project Property Settings frame
- **Type**: Initial Head = 0 in the Default Project Property Settings frame
- **Type**: $S_s (1/m) = 1E-05$ in the Default Project Property Settings frame
- **Type**: $S_y = 0.1$ in the Default Project Property Settings frame
- **Type**: Name = Salt in the Species Parameters table
- **Type**: Description = Salt in the Species Parameters table
- **Type**: $DRHODC = 0.7$ in the Species Parameters table
- **Type**: $MDCOEFF = 0.308$ in the Species Parameters table

**Note**: $MDCOEFF$ is the species-specific molecular diffusion coefficient. This parameter is only used if the multi-diffusion option is set to true in the DSP package advanced settings. While we have entered a value here, it will not be used. We will specify a model-wide molecular diffusion coefficient at a later step.
Now that the modeling objectives are defined you may advance to the next step of creating the numerical grid. Click the white right arrow in the blue circle at the top of the Numerical Model window to advance to the Define Numerical Model step.

- Click [ ] (Next Step) to proceed to the Define Numerical Model workflow step
- Click [Yes] to dismiss the warning message about the model start date
- Now click on the [Create Grid] icon

**Define Model Grid**

We will now proceed with the model grid creation. You should now see the Create Grid workflow step, as shown in the image below:
We will create a grid with 1 row, 44 columns and 27 layers. Specifying the number of rows/columns and the overall grid extents horizontally will be quite easy, as these values can be specified directly in this window. To specify layer elevations for all 27 layers would be quite tedious though, so we will initially create the grid with a single layer with the elevations being the maximum and minimum final layer elevations. We will then refine the initial layer into 27 equally thick layers.

Please input the number of Rows and Columns and change the number of layers to 1. The top elevation of the first layer shall be set to 6 m, and the bottom elevation to -156 m.

- **Rows** = 1 in the **Grid Size** frame
- **Columns** = 44 in the **Grid Size** frame
- **X Min** = 0 in the **Grid Extents** frame
- **X Max** = 600 in the **Grid Extents** frame
- **Y Min** = 0 in the **Grid Extents** frame
- **Y Max** = 13.636 in the **Grid Extents** frame
- **Cell width** = calculated this value can be adjusted as per project requirements
- **Cell height** = calculated this value can be adjusted as per project requirements
- **Number of Layers** = 1
- **Layer1 - Top Elevation** = 6
- **Layer1 - Bottom Elevation** = -156
The screen should now look like the image below:

- [Create Grid] button at the top-left of preview window

The Model Explorer (the lower left section of the interface) will be populated with the newly created model structure items. You will also see 2 new surfaces created in the Data Tree. These surfaces represent the top and bottom surfaces of the model layers and are consequently transformed automatically into ‘Horizons’ in the Model Explorer. Each pair of horizons in turn defines a zone, so you will also notice one new zone in the Model Explorer.

Your grid has been created and you’ll notice that the Properties section of the model structure is empty, so the next step will be to define boundary conditions and the model properties. The ‘View/Edit Grid’ portion of the workflow allows the user to make any necessary changes to the grid structure. This will be particularly useful for us, since we need to refine our initial grid vertically to have 27 layers (as defined by the original Elder model).

- Click [Next Step] to proceed to the View/Edit Grid step

In the middle of the interface you will see a Toolbox. The buttons under this toolbox allow you to edit the grid in the horizontal direction (i.e. Edit Grid...), to edit the grid in the vertical direction (i.e. Edit layers...), to extract a subgrid from your model (i.e. Create subgrid...), or
to assign model cells as active/inactive (i.e. the Inactive Cells frame). We will utilize the Edit layers functionality to refine our initial single layer grid, and then use the Inactive Cells feature to deactivate several flow cells in the top model layer.

- Click the [Edit layers…] button under the Toolbox
- Type: FinalGrid in the New grid name field
- Type: 27 in the By a factor of field
- Click the [Apply edit] button at the bottom left
- Before finalizing the layer edit by clicking OK, the Edit layers window should look like the image below.
- Finalize the layer edit by clicking [OK]

Once again you should see that the Model Explorer will be populated with the newly created model structure items, and a new workflow window will open. We will continue working with the FinalGrid workflow, so feel free to minimize the initial grid in the Model Explorer, and close the initial workflow window:
The final change to the grid is to specify two inactive areas in model layer #1. To make this change we will activate the **Row** view (Row 1) and assign inactive cells using the polyline drawing function:

- **Row** in the **Views** frame, to activate Row view
- **Layer** in the **Views** frame, to deactivate Layer view
- **Exaggeration**: 5 in the field above the viewer to decrease the vertical exaggeration
- **[Assign]** > **[Using Polyline]** button under the Toolbox, under Inactive Cells
- Draw a polyline in layer 1, from Column 1 to 11 (click once to start, double-click to finish)
- Draw another polyline in layer 1, from Column 34 to 44 (click once to start, double-click to finish)
- **[Finish]** button under the Toolbox
- **[OK]** in the **Copy to Row(s)** window that appears
- The final grid should look like the image below:
We have now recreated the grid as required by the Elder model and are ready to proceed to the next step in the workflow, which is to define property values.

- Click [Next Step] to proceed to the Define Properties step

Define Properties

You should now be viewing the Define Properties workflow step. You may want to inspect that the model properties are set according to the default property values that you defined during the Define Modeling Objectives step. Select any of the properties from the toolbox and click on the Edit… button, and you will see the values assigned to each cell of the model for the respective property. For example, the image below displays the Edit Property window for the conductivity property group:

- Click [Edit...] in the Toolbox to open the Edit property window for a given property group
• Click [OK] in the dismiss the **Edit property** window

Remember that at any step during the numerical workflow you have the possibility to inspect your grid in four different views: plan, two cross-section views and a 3D view. These views can be toggled on and off by checking the respective check-boxes in the **Views** area. Make sure you are looking at Row 1 in the Layer View window. If you have more viewer windows open, make sure that Row View is the active window, as shown in the image below:

Now we will assign property values as shown in the following image:
We will begin with the conductivity values for the top and bottom model layers:

- Select [Conductivity] from the first menu in the Toolbox frame
- Click [Assign] > [Polyline] buttons under the Toolbox
- Type: 5 in the [Exaggeration] field above the viewer to decrease the vertical exaggeration
- Draw a polyline across all of layer 1 (click once to start, double-click to finish)
- Draw another polyline across all of layer 27 (click once to start, double-click to finish)
- Click [Finish] button under the Toolbox
- The New Property Zone window will open.

In this window we will assign the selected cells to a New property zone, and apply the desired Kx, Ky and Kz values (i.e. 1E-05 m/day).

- [New] button at the top-left of the window, to create a new property zone
- Type: 1E-05 in the Value field for Kx, Ky and Kz
- [OK] to accept inputs and create the new property zone
- The New Property Zone window should look like the image below before clicking [OK]:
We will repeat this process for initial concentrations, assigning a value of 285,700 mg/L for layer 1 ONLY.

- Select [Initial Concentration Salt] from the first menu in the Toolbox frame
- Click [Assign] > [Polyline] buttons under the Toolbox
- Draw a polyline across all of layer 1 (click once to start, double-click to finish)
- Click [Finish] button under the Toolbox
- The New Property Zone window will open
- Click the [New] button at the top-left of the window, to create a new property zone
- Type: 285700 in the Value field for Concentration
- [OK] to accept inputs and create the new property zone

The remaining changes to property values will be made to the entire model, so we can use the Edit function rather than assigning new property zones using the drawing tools. We will update the effective and total porosity values (through the Storage property group) and longitudinal dispersivity (through the Dispersion property group).

- Select [Storage] from the first menu in the Toolbox frame
- Click [Edit...] button under the Toolbox
- Type: 0.1 in the Effective Porosity column, then use [F2] to propagate through all cells
- Type: 0.1 in the Total Porosity column, then use [F2] to propagate through all cells
- [OK] to accept inputs and create the new property zone

- Select [Dispersion] from the first menu in the Toolbox frame
- Click [Edit...] button under the Toolbox
- Type: 0 in the Longitudinal dispersivity (m) column, then use [F2] to propagate through all cells
• [OK] to accept inputs and create the new property zone

Finally, one final change must be made to the model level dispersion parameters. These can be accessed by right-clicking the Dispersion property group under the Model Explorer and selecting Dispersion Parameters as shown in the image below:

- Right-click Dispersion from the Model Explorer (under FinalGrid1 > Run1 > Inputs > Properties > Transport)
- Click [Dispersion Parameters] from the menu that appears

The Dispersion Parameters window will open

- Type: 0.308 in the Diffusion Coeff. (m^2/day) field, then use [F2] to propagate through all cells
- Click [OK] to confirm and assign values
Note: this diffusion coefficient is the model-wide molecular diffusion coefficient. This is different than the species-specific diffusion coefficient applied during the Define Modeling Objectives step. The species-specific value is only used if the multi-diffusion option is set to true in the DSP package advanced translation settings.

Now is a good time to save the project.

- Click [File] > [Save Project] from the main menu

Define Model Grid

If you are satisfied with the property values assigned to the model, advance to the Define Boundary Conditions step in the numerical workflow. We will assign the model boundaries in Row view.

- Click [Next Step] to proceed to the Define Boundary Conditions step

At this stage we will assign boundary conditions to our model, starting with a constant head boundaries at the top-left and top-right corners of the model (see the figure at the very beginning of this tutorial). We will do this by simply selecting the two desired cells.

- [Constant Head] from the first menu under the Toolbox
- [Assign] > [Cells] from the menu under the Toolbox
- Left-click once in Layer 2, Row 1, Column 1
- Left-click once in Layer 2, Row 1, Column 44
- Click [Finish] under the Toolbox
- The Define Boundary Condition window will open
Click [Next>>] to accept default name and proceed

We will assign a constant head value of 0 m to both of the selected cells. For both cells, type a value of 0 in the starting and ending head fields, select **Specified Density** as the density option, and apply a density of 1000 kg/m$^3$.

- **Type: 0** in the **Starting Head (m)** field, for both cells/rows
- **Type: 0** in the **Ending Head (m)** field, for both cells/rows
- **Select Specified Density** in the **Density Option** field, for both cells/rows
- **Type: 1000** in the **Density (kg/m$^3$)** field, for both cells/rows

Click the [Finish] button to finalize and create the constant head boundary condition

You will notice that both of the selected cells will be populated with red dots. These red dots represent the newly created boundary condition. Boundary conditions within Visual MODFLOW Flex are color-coded for quick identification.

We will now apply two constant concentration boundary conditions. In the top layer we will apply a high salt concentration (285.7 kg/m$^3$), and in the bottom layer we will apply a constant concentration of 0 kg/m$^3$. This concentration gradient is what will drive water flow and solute transport in this model.

- **[Constant Concentration]** from the first menu under the **Toolbox**
- **[Assign] > [Polyline]** from the menu under the **Toolbox**
- **Draw a polyline across active zone of layer 1 (click once to start, double-click to finish)**
- **Click [Finish] under the Toolbox**
- The **Define Boundary Condition** window will open
- **Click [Next>>]** to accept default name and proceed
• **Type:** 285700 in the Salt (mg/L) field, then use [F2] to propagate through all cells
• Click the [Finish] button to finalize and create the constant head boundary condition

**Note:** if the drawn polyline extends into the inactive zone, you may see a **Resolve validation failures** window appear. In this case it is simply identifying that some of the selected cells are in the inactive zone, and the boundary conditions for those cells can either be deleted or the cells can be reactivated. The default solution will be to **Delete invalid cells.** If you **DO** see this window, simply click **Apply** then **Close** to resolve potential validation errors.

Now we will repeat the steps above to apply a constant concentration of 0 mg/L to the lowest model layer.

• **[Constant Concentration]** from the first menu under the **Toolbox**
• **[Assign] > [Polyline]** from the menu under the **Toolbox**
• **Draw a polyline across active zone of layer 27** (click once to start, double-click to finish)
• Click [Finish] under the **Toolbox**
• The **Define Boundary Condition** window will open
• Click [Next>>] to accept default name and proceed
• **Type:** 0 in the Salt (mg/L) field, then use [F2] to propagate through all cells
• Click the [Finish] button to finalize and create the constant head boundary condition

Now all of the desired boundary conditions have been applied across the entire model domain. Your display should look something like the image below. Note the beige dots in layers 1 and 27, indicating the presence of the constant concentration boundaries. Also note the red dots indicating the constant head boundaries in layer 2 (top-left and top-right). Finally, note how each new boundary condition has been added into the **Model Explorer.**
We now have enough data to run this density-driven flow model, but first we have to define our model translation settings.

Translate and Run the SEAWAT Model

Navigate to the Select Run Type > Single Run step in the numerical workflow. You may click directly on this step as it is colored green, or you can navigate there step-by-step using the navigation arrows at the top of the workflow window.

- Click [Single Run] from the Workflow Navigator, to proceed directly to the Single Run workflow step

You will be prompted to select which engines to run. By default, USGS SEAWAT from WH should already be selected. This is because this is the only flow engine within Visual MODFLOW Flex that allows you to model density-dependent flow. If you click on the Flow Engine menu you will see that USGS SEAWAT from WH is the only available option. The engine settings should look like the image below:
You will now advance to the **Translate** step of the workflow. This step will translate all the input data defined for the model into “packages” (i.e. text files) that can be run by SEAWAT.

- Click [Next Step] to proceed to the **Translate** step

The translation settings will appear; this allows you to adjust solvers and their parameters (number of iterations, head-change criterion, damping factors), package settings, output control, etc.. In the **Settings** section of the SEAWAT branch you can change the **Steady-State Simulation Time** to 7300 by entering this value directly. Select **Geometric Multigrid Solver (GMG)** as the solver in the **Solvers** section. We will also decrease the **Head change criterion (HCLOSE)** and **Residual criterion (RCLOSE)** to 0.001.

- Go to the [SEAWAT] > [Settings] section of the menu
  - Type: 7300 in the **Steady-State Simulation Time** field

- Go to the [SEAWAT] > [Solvers] section of the menu
  - Select **Conjugate Gradient Solver (PCG)** in the **Selected Solver** field
  - Type: 50 in the **Max. Outer Iterations (MXITER)** field
  - Type: 25 in the **Max. Inner Iterations (ITER1)** field
  - Type: 0.001 in the **Head Change Criterion (HCLOSE)** field
  - Type: 1 in the **Residual Criterion (RCLOSE)** field
  - Type: 5 in the **Printout Interval (IPRPCG)** field

- Go to the [SEAWAT] > [Layers] section of the menu
  - Apply layer type ‘0: Confined, constant S,T’ for ALL layers

- Go to the [SEAWAT Transport] > [General] section of the menu
  - Select **Total** in the **Porosity Options** field
  - Type: 0.1 in the **Courant Number** field

- Go to the [SEAWAT Transport] > [Solution Method] section of the menu
  - Select **Central Finite Difference (CFD)** in the **Advection Method** field
  - Select **Yes** in the **Use Implicit GCG Solver** field
  - Type: 1E-08 in the **Relative Convergence Criterion** field
- **Type: 5** in the Concentration change printing interval field
- **Type: 3** in the Initial step size (DT0)(days) field
- **Type: 3** in the Max step size (days) field

- Go to the [SEAWAT Transport] > [Output Control] section of the menu
- **Type: 7300** in the Simulation time length (project time units) field
- **Type: 3000** in the Max number of transport steps field

- Click [Add row] x6 press the Add row button six times to specify six output times
- **Type: 365** [press Tab], **730** [press Tab], **1095** [press Tab], **3650** [press Tab], **5475** [press Tab] and **7300** as the desired output times in each row

The solver settings window for the **Output Control** section should look like the image below:

![Solver Settings Window](image)

It is also important to know that individual settings for the SEAWAT variable-density flow (VDF) and viscosity (VSC) packages can be set through the **SEAWAT Transport > Advanced Settings** menu.

The VDF translation menu allow you to specify how groundwater densities are calculated using the **Density Option** setting (i.e. using salt only, using multiple species, or using user-specified densities), how internodal density is calculated (i.e. central-in-space or upstream-weighted algorithm), whether a variable density water table correction is applied or not, and to specify constants such as minimum/maximum fluid density, and a reference fluid density. If you access the VDF translation settings menu you should see the following (**no changes required**):
The VSC translation settings allow you to specify how groundwater viscosities will be calculated in the simulation. The Viscosity Option setting allows you to choose whether viscosity is calculated using salt only (no temperature dependence), using temperature and additional species (e.g. salt), or whether specifies viscosities will be used. You can also specify constant values such as maximum, minimum and reference viscosities in this menu. If you access the VSC translation settings menu you should see the following:
However, we do not actually want to run the .VSC package in this simulation, assuming that there is no viscosity effects at higher concentrations. To remove the .VSC package from this model we will change both the ‘Run’ and ‘Translate’ fields to ‘No’.

- Select [No] in the Run field
- Select [No] in the Translate field

At this time the desired translation settings have all been specified and we can proceed to translating the model input files. Press the translate button to begin the translation process. As soon as you click the translate button Visual MODFLOW Flex will begin translating all the required input files for a SEAWAT model.

- Click [ ] to create the SEAWAT input packages. Check the log to confirm the translation has finished.
- Click [Yes] to dismiss the warning about initial heads

A translation log will run for a few moments and should terminate with the following message:

################################ Translation Finished ################################

You should also see a series of new tabs at the top of the translation window, corresponding to the various input files which have been generated. You can click these tabs to view the contents of each input file. You can also click the open file button [ ] to open the folder containing the input files. If you aren't happy with the translated files you can always return to the Settings tab, make changes and retranslate the files. For this model run you won’t need to make any further changes.
You may now proceed to the **Run Numerical Engines** step.

- Click [Next Step] to proceed to the **Run** step
- Click [Run] to run the SEAWAT model

The numeric engines will start running and display progress in the main window. SEAWAT will run first followed by any secondary engines selected during the **Single Run** workflow step (e.g. MODPATH or ZoneBudget; we haven’t selected any additional engines in this model). Each engine will have an information window that displays simulation results and progress. Clicking on the tab of the respective window will enable you to view detailed results of each run. When your model is finished running your results screen should look like the image below:
If you would like to inspect a record of the run process you may open the listing (.LST) file by clicking on the **Open Engine Files…** button at the top of the window. The model run may take several minutes to complete, since SEAWAT models are more complex than most contaminant transport models. A SEAWAT model may take 2-4 times longer to run than a MODFLOW model of the same dimensions due to the coupling of flow and transport processes.

**Output Visualization**

To see the results of the simulation navigate to **View Results -> View Maps** in the workflow tree. Equipotentials will be shown in plan view by default. You may want to change your map appearance by using the icons at the top of the window and/or right clicking on Heads in the **Model Explorer** and select **Settings…**. To see your equipotentials and the water table in cross-section activate the **Row** viewer (put a check-mark) and select the row number along which you would like to inspect the results.

- Click [ ] (Next Step) to proceed to the **View Results** step
- Click [View Maps] to proceed to the **View Maps** workflow step
- ✅ **Row** in the **Views** frame, to activate Row view
- ❔ **Layer** in the **Views** frame, to deactivate Layer view
- Activate ✅ **Water Table** under the Outputs/Flow section of the **Model Explorer**

Depending on your settings, you should see something like the figure below:
Note: you can maximize the viewer window by deactivating the Model Explorer and/or Workflow Navigator panels. Use the toolbar buttons \[
\begin{align*}
&\text{on} & &\text{off}
\end{align*}
\] to turn these panels on/off.

To better visualize the flow field in the model domain you can activate the **Velocity** outputs under the Model Explorer. The velocity flow field indicates that the diffusive flux of salt in the model domain is driving water flow in two recirculation cells, with the greatest flow velocities in the middle of the model and circulating upward toward the edge of the model where the constant heads boundaries have been applied.

- Activate \(\mathcal{Z}\) **Velocity** under the Outputs/Flow section of the **Model Explorer**
Now you may activate the concentration results, which will display the diffusion of salt throughout the model domain at different output times. While using the Flex/composite viewer, activating the concentration results will automatically deactivate the head equipotential results. This is because only one set of 3D gridded data may be displayed in the Flex/composite viewer at any given time. We can use a 3D window to display multiple sets of data at the same time.

For now, simply activate the concentration results:

- Activate ☑ Concentration (Salt) under the Outputs/Flow section of the Model Explorer
Note: if the contours don’t match the figure above initially try right-clicking ‘Concentration (Salt)’ from the Model Explorer, accessing the contour line settings and updating the minimum contour level to 0 mg/L.

You can use the time-step picker buttons above the Flex viewer to display the concentration results at various output times. If you update the viewer to display the results at day 365 you should see something like the image below:

- Press [ ] under the Outputs/Flow section of the Model Explorer
Using the cell-inspector tool, review salt concentrations throughout model layer 2 and you should notice some unusual results.

- Press [ ] under the Outputs/Flow section of the Model Explorer

The Cell Inspector window will open, allowing you to select which information will be displayed on a cell-by-cell basis. We’re only interested in the results of the model run, so we can deactivate the position, properties, and boundary conditions categories:

- ? Position to turn off position information
- ? Properties to turn off properties information
- ? Boundary Conditions to turn off boundary condition information
- ? Budget turn off the flow budget information
- Click once on Layer 2, Row 1, Column 11

The Cell Inspector window will now display the results of the simulation for the selected cell, as shown in the image below. You can see from these cell inspector results that the salt concentration in cell 2, 1, 11 is \(-43,772 \text{ mg/L}\). These results clearly include non-physical values (e.g. negative concentrations) and can be attributed to truncation errors and/or
numerical dispersion associated with the selected solution method for the transport equation (i.e. Central Finite Difference method).

You can improve the results by adjusting the solution method inputs. Return to the Translate workflow step and make the following changes:

- Click [Translate] directly from the workflow navigator, to return directly to this workflow step
- Go to the [SEAWAT] > [Solvers] section of the menu
- Type: 1E-7 in the Head change criterion (HCLOSE) field
- Type: 0.01 in the Residual criterion (RCLOSE) field

- Go to the [SEAWAT Transport] > [Solution Method] section of the menu
- Select Total Variation Diminishing (TVD) in the Advection Method field
- Type: 20 in the Max. number of outer iterations field
- Type: 100 in the Max. number of inner iterations field
- Type: 1E-09 in the Relative Convergence Criterion field

Now re-translate and re-run the model

- Click [Translate] directly from the workflow navigator, to return directly to this workflow step
- Click [No] to dismiss the warning(s) about No package data
- Click [Yes] to dismiss the warning about initial heads
- Click [(Next Step)] to proceed to the Run step
• Click to run the SEAWAT model

When the model is finished running you should see the following results for salt concentrations at day 365 and 7300, respectively. You can also double-check salt concentrations throughout layer 2 to ensure that no negative concentrations are present:
Compare the contours at day 365 and 7300 to the following published results (Guo and Langevin, 2002):
Now is a good time to save the project. Click [File]→[Save Project] from the main menu.

*****This concludes the 'SEAWAT - Elder Model' tutorial.*****

3.5 PEST with Pilot Points

This section provides instructions on using Visual MODFLOW Flex to setup, run, and interpret a Parameter Estimation/Predictive Analysis simulation. In addition, this tutorial provides a brief description of the input parameters and settings required by PEST. A detailed
description of the algorithms, parameters, input files, and other options for PEST are available in the PEST User Documentation. This can be accessed from www.PESTHomepage.org

This exercise demonstrates some of the advanced and exciting opportunities for model calibration and uncertainty analysis using PEST in Visual MODFLOW Flex. This exercise is based on the problem described in "Using Pilot Points to Calibrate a MODFLOW/MT3D Model", by John Doherty (Watermark Numerical Computing), and has been adjusted to work with the PEST workflow inside Visual MODFLOW Flex.

Objectives

By the time you have finished this exercise you will have:

- used pilot points as a means of characterizing the spatial distribution of an aquifer hydraulic property;
- used PEST’s advanced regularization functionality in conjunction with geostatistically-based regularization constraints;
- used the Visual MODFLOW Flex GUI to:
  - Build input files for PEST
  - Run PEST
  - Analyze the results
  - Save adjusted parameters as new model inputs

Before you start

You are encouraged to familiarize yourself with the concepts and applications of PEST prior to using in Visual MODFLOW Flex. The time spent on this will make your experience with parameter estimation much more productive, and will likely help you to overcome any difficulties you may experience the first time you run PEST.

In addition, if you are not familiar with the Visual MODFLOW Flex graphical environment, please take a few minutes to review the Program Overview section.

Note:

You must have a Pro or Premium license in order to use the PEST module in Visual MODFLOW Flex.

Required Files

Several files are required for this exercise, which should be included with the Visual MODFLOW Flex installation.

These files are available in your public "My Documents" folder, typically:
C:\Users\Public\Documents\VMODFlex\Tutorials\PEST

If you cannot find these files, please download the PEST Tutorial project from our website, and unzip to a desired folder on your computer.

Introduction

- Launch Visual MODFLOW Flex
- Click [File]-[Open Project]
- Navigate to your Public 'My Documents' folder, and locate "Documents\VMODFlex\Tutorials\PEST", and open the 'pest-tutorial.amd' file
- Click [Open]
- In the Numerical Workflow window, locate the workflow tree (this is shown on the left in the window below)
- If you're not already on this step, click on the 'Select Run Type' in the numerical workflow tree.
- Click on the [PEST Run] button

A new PEST Workflow window will load as shown below

Please Note: if this does not appear right away, select this tab from the list of workflows across the top of the window, highlighted in image below)
Define Observations

- The first step is to define the observations you want to include for calculating the objective function, and assign weights to various observations.
- In this example, there are 21 observation wells where heads were measured at several intervals.
- All head locations and associated time varying heads are selected by default, and will be used with a default weight = 1.
- No changes will be needed for this exercise

Define Property Parameters

- Click [Next Step] to proceed to the 'Define Property Parameters' step.
- The 'Define Property Parameters' window will then appear:
- At this step, select which parameters you want to include in the PEST run.
- In the table at the top, select which property parameters you want to include; in the table at the bottom, select which property zones you want to include. For this exercise, all Kx property zones will be included (these are selected by default).
- For each parameter, you can specify to "Tie" it to another parameter. You can also specify the Transformation option (by default, all Conductivity parameters are set to Log transformation).
- In the table at the bottom, for each property zone, you must specify minimum and maximum values; defaults are provided.
- In addition, the value from each zone is also displayed to assist in defining reasonable minimum/maximums.
- Enter 1 for the Minimum for each zone.
- Enter 300 for the Maximum for each zone.
Click [Next Step] (Next Step) to proceed to the 'Define Pilot Points' step.

Define Pilot Points

The next step is to 'Define Pilot Points' as shown below
Pilot points are XY points with an initial value for each parameter. Pilot points can be imported from .TXT file, XLS, .SHP file, or assigned manually by digitizing in the 2D environment. An example of the pilot points on top of conductivity zones is shown below.
The general steps are as follows:

- Select **pilot-points-zone1** data object from the Data Explorer (tree)
- Click (Insert button) at the top of the 'Define Pilot Points' window to add these points.
- Select which parameter zones the points represent under the Parameter Zones grid in the upper right section of the window.
- For this set of pilot points, **Kx-Zone1** is fine (this should be selected by default)
- Your display should now appear as shown below:
Now repeat these steps to define the pilot points for Kx Zone2

- Select 'pilot-points-zone2' data object from the Data Explorer (tree)
- Click (Insert button) at the top of the 'Define Pilot Points' window to add these points.
- Select Kx-Zone2 for these pilot points (if it is not already selected)

Repeat these steps for Kx Zone3 and Zone4

- Select 'pilot-points-zone3' data object from the Data Explorer (tree)
- Click (Insert button) at the top of the 'Define Pilot Points' window to add these points.
- Select Kx-Zone3 for these pilot points
- Select 'pilot-points-zone4' data object from the Data Explorer (tree)
- Click (Insert button) at the top of the 'Define Pilot Points' window to add these points.
- Select Kx-Zone4 for these pilot points
- When you are finished, your display should appear similar to the figure below.

In the table at the bottom of the window, you can adjust some parameters for specific pilot points. The main use of this table is to specify which pilot points (if any) are Fixed (Hard) and their initial values. Fixed pilot points are those locations where you are confident in the measured parameter value (e.g., pumping/slug test locations), and you want these values to remain Fixed during the PEST run.

Above this table, there is a combo box where you select which parameter zone should be shown. VMOD Flex allows you to combine multiple pilot point sets (e.g., hard/soft) for parameter zones.

In this example, leave the defaults as is.

- Click (Next Step) to proceed to the Define Kriging Variograms.

**Define Kriging Parameters**

The use of pilot points in characterizing the spatial distribution of a hydraulic property must be accompanied by a mechanism whereby hydraulic property values assigned to pilot points are spatially interpolated to the cells of the finite difference grid. Spatial interpolation is accomplished using the Kriging algorithm. Kriging is a method of spatial interpolation based
on geostatistics. The cornerstone of geostatistics is the variogram. A variogram describes the extent to which hydraulic property values (or any other type of data) pertaining to any two points are likely to be different from each other as a function of the distance between those points.

One of the benefits of using Kriging as a basis for spatial interpolation is that the factors by which hydraulic properties at pilot points are multiplied before summation to obtain the hydraulic property value at a particular grid cell are independent of the actual hydraulic property values at the pilot points. Hence a set of “Kriging factors” pertaining to each of the cells of the finite difference grid can be calculated in advance of the actual interpolation process. Since the interpolation process is undertaken again and again as the model is run repeatedly by PEST, the fact that it is not necessary to repeat calculation of the Kriging factors on each occasion that the model is run can result in large savings in the time required to complete the overall parameter estimation process.

In this example, there are 4 hydraulic property zones as shown below:

- Zone 1: river alluvium
- Zone 2: creek alluvium
- Zone 3: western basalt
- Zone 4: eastern basalt

Each zone is represented by a geostatistical structure. Each of these structures cites one variogram (though it could cite up to five). “Structure1” will be used to characterize zone 1 of our model domain (i.e. the river alluvium), “Structure2” will be used to characterize zone 2 (i.e.
the creek alluvium) whereas “Structure3” will be used to characterize zones 3 and 4. Note that the variogram assigned to these latter zones is quite unimportant; because there is only one pilot point assigned to each of them, all cells within these zones will be assigned the one interpolated value (same as the respective pilot point) irrespective of the variogram.

In VMOD Flex, a default variogram is generated for each parameter zone, with type 2 (Exponential).

However, the structures should be modified.

- Click 'Zone1' under 'Kx' in the tree. The display will appear as below:
In the editor on the right side of the display, for 'Transform', set this to log.

Repeat these steps, setting log 'Transform' for Kx in Zones 2, 3, and 4.

Thus any variogram cited in each of these structures must pertain to the spatial distribution of the logarithm of the pertinent hydraulic property. This is in accordance with the fact that most studies cited in the groundwater literature which treat transmissivity and/or hydraulic conductivity as a regionalized variable indicate that its distribution is better described by a log variogram than by a variogram based on native property values.

For this example, the default parameters for the variograms (all will use Exponential) is sufficient. However, for Zone2, we will define a value of 2.0 for the Anisotropy, with the direction of anisotropy coinciding with the direction of the creek. Alignment in the direction of the creek is based on the premise that channel structures within this old creek valley will make it more likely for hydraulic property similarity to prevail in this direction than in a direction at right angles to it.

The variogram parameters can be adjusted by selecting a Variogram from the tree as shown below.

- Click 'Variogramkx2' under the Variograms node in the tree.
- Enter 2 for the 'Anisotropy'.
• Enter 45 for the ‘Bearing’. This value allows you to make the variogram anisotropic in a certain direction; ‘Bearing’ is an angle of rotation.
• The display should appear as below:

![Image of variogram settings]

• Click [Next Step] (Next Step) to proceed to the Select Run Type.

Select Run Type

At this step, choose the type of PEST Run. If you want to run PEST, then some additional options will be needed, such as define regularization and adjust the PEST control file. If you want to run Sensitivity Analysis, this can also be launched.

![Image of run type selection]

• Click the [Parameter Estimation] button from the main window, as shown above.
• The next step will be to choose the Regularization options.
Select Regularization

At this step, you can choose the type of Regularization to run. Regularization is a process whereby additional information is introduced into the objective function of the model in the form of ‘prior information equations’. These prior information equations can take many forms.

Tikhonov Regularization can induce PEST to prefer a more homogeneous solution (i.e. pilot points that are near one another should have similar values) or Tikhonov Regularization can induce PEST to prefer parameter values which are as close to their initial values as possible. The Tikhonov regularization method adds more information to the overall problem, but the additional information helps to guide PEST toward acceptable parameter estimates as informed by your knowledge of the modeled area and to avoid ‘overfitting’ solutions.

Another method of regularization supported in Visual MODFLOW Flex is singular value decomposition (SVD) regularization. The SVD process achieves numerical stability by subtracting parameters and/or combinations of parameters from the calibration process. As a result of the subtraction, the calibration process is no longer required to estimate either individual parameters or combinations of correlated parameters that are inestimable on the basis of the calibration dataset. These combinations are automatically determined through singular value decomposition (SVD) of the weighted Jacobian matrix.

For this exercise we will not perform any regularization.

- Click [No Regularization] button from the main window
- The next step will be to adjust the PEST control file.
Edit PEST Run Settings

The last step before running PEST is to view and adjust the PEST Control file.

If you are familiar with the PEST file format/structure, you can adjust the PEST Control file in this window, or copy into a text editor, make changes, and paste the adjusted contents back in this window. A full explanation of the PEST control file is available in the PEST manual (http://www.pesthomepage.org/getfiles.php?file=pestman.pdf)

For this exercise, the default values are fine.

Before starting the PEST Run, it is a good idea to check the PEST Input files. PEST provides a utility to do this, called PESTCheck.
Click the [ ] button on the workflow toolbar
You should receive a confirmation that no errors were found.
Click [OK] to proceed.
Click [ ] (Next Step) to proceed to the Run PEST window.

Run PEST

Click [ ] (Run PEST) to start the PEST Run.

The pest.exe will load in a DOS command window, and show the progress as seen below.

Depending on the speed of your computer, the PEST run should take between 3-5 minutes. As PEST runs, you should see the objective function (phi) decreasing over each optimization iteration; pay attention to these values in the DOS window. PEST will run a total of 25 optimization iterations and a total of 1366 model runs. PEST should reach a final objective function (phi) value of approximately 2.06 E-02.

When PEST finishes, you should see a confirmation message in the main window, below the ‘PEST Run Log’ tab, as shown below.
After the PEST run completes, you can analyze the results.

- Click (Next Step) to proceed to the ‘Analyze Results’ step.

**Analyze Results**

VMOD Flex presents the results of the PEST run, with one tab per output file.
- **Record file (.REC):** contains parameter values, objective function, sensitivities, etc..
- **Sensitivities for Observations (.SEO):** contains the observed and simulated values with the sensitivities
- **Sensitivities for Parameters (.SEN):** provides the information on parameter sensitivities
- **Residuals (.RES):** contains the adjusted calculated vs. observed values and residuals

The results from these files can be exported into Excel for charting.

- Click ![Export](image) to export the PEST results to an Excel spreadsheet.

A window will appear which allows you to select the location of the exported Excel spreadsheet, and to select from available templates. Two templates are available; one which will export the available PEST data to a series of worksheets/tables, and a 2nd template which will also generate a number of charts for each type of PEST data.
• Select the 'pest-results-chart-and-table.xlsx' template
• Click the checkbox besides 'Open file when finished'
• Select an output location
• Click Export; the Export Pest Results window should look like the image below before exporting:

An Excel spreadsheet will open which summarizes the results of the PEST run. Several data tables and charts are included; spend a few minutes reviewing them. The screenshots below display a small selection of the available PEST data. For more information about the PEST results please visit the PEST website (link provided at the beginning of this tutorial).
If the results look reasonable, you can save the adjusted Kx parameter zonation as inputs for a new model; this is explained in the next section.

- Click [Next Step] to proceed.

**Save PEST Parameters as New Inputs**

After reviewing the PEST output, if the adjusted parameter values seem reasonable, you can save these parameters as inputs for a new model run.
• Click on the [Create New Model Run with PEST Results] button.

VMOD Flex will save the adjusted model parameters in a new model run within the same project. This new model run will appear in the Model Explorer below the most recent model run. A new workflow window will also appear with this model run.
• Click on 'Define Properties' from the workflow tree (if not already selected)
• From the Toolbox, select 'Kx' as shown below; you should then see a color flood of the Kx values.
You can mouse over the 2D display to see the range of Kx values. Click on the button on the toolbar to show color shading with contour lines.

You must Translate and Run this new model run in order to see the updated MODFLOW results using the adjusted Kx parameters from PEST.

****This concludes the 'Model Calibration Using PEST with Pilot Points' tutorial.****

3.6 MODFLOW-USG Tutorial

The following example is a walk through of creating a MODFLOW-USG groundwater flow model. The exercise is based on a conceptual model project that is already defined in one of the Tutorial projects.

Objectives

- Open and review a VMOD Flex project that contains a conceptual model
- Define an UnStructured Grid, with refinement around the wells and boundaries
- Convert the conceptual model to a numerical model
• View the corresponding flow properties for the MODFLOW-USG model
• View the boundary condition cells for MODFLOW-USG model
• Translate and Run MODFLOW-USG
• View and Analyze the Results
• Understand how Visual MODFLOW flex accommodates inputs and outputs for a MODFLOW-USG model

Pre-requisites

This tutorial assumes that you are familiar with the Visual MODFLOW Flex environment. If you are new to Visual MODFLOW Flex, please take a few minutes to review the Program Overview section, and turn through the Conceptual Modeling tutorial prior to starting MODFLOW-USG.

Opening the Project

• Launch Visual MODFLOW Flex
• Click [File]-[Open Project...]
• Navigate to your 'Public Documents' folder then locate 'VMODFlex\Tutorials\MODFLOW-USG\Transient'
• Open this folder, select the 'usg-transient.amd' file.
• Select [Open]
• The project will load.

⚠️ Please Note: if you are unable to locate the project for this tutorial, you may download a copy from our website

Review the Conceptual Model

The surficial geology at the site consists of an upper sand and gravel aquifer, a lower sand and gravel aquifer, and a clay and silt aquitard separating the upper and lower aquifers. These are defined as three property zones in the conceptual model, with hydraulic conductivity for the aquifer as 2E-04 m/s and the aquitard having a horizontal conductivity of 1E-10 m/s and vertical conductivity (Kz) = 1E-11 m/s.

Initial heads are defined using the Ground surface, and this value is applied to all layers.

The boundary conditions consist of a constant head in upper aquifer, in the north part of the model, a river along the southern edge, and two pumping wells screened over the bottommost property zone, each pumping 400 - 700 m3/day, with a time-varying pumping
schedule. Constant head values are also defined in the lower aquifer. Each boundary condition consists of a few simple stress periods to illustrate a transient MODFLOW-USG model run.

When discussing the site, in plan view, the top of the site will be designated as north, the bottom of the site as south, and the left side and right side as west and east, respectively. Groundwater flow is from north to south (top to bottom).

Define an UnStructured V-Grid

- Select the 'Conceptual Model 2' tab (from the list of active workflows/windows, at the top of the display)
- Navigate to the 'Select Grid Type' step in the workflow (if its not already selected), as shown below.

- Click on the 'Define UnStructured V-Grid' button, and the following window will appear:
The first step in defining the Unstructured Grid is to provide the desired shape add-ins. These are polylines, polygons, and points within the model boundary. The grid generator in VMOD Flex will use these as control points (grid generator nodes) during creating the unstructured grid. By default, the Add-ins List contains the model boundary (polygon) and any linear or point boundary condition currently defined for the conceptual model. Additional add-ins may be added to the list using data objects from the Data Explorer, and then clicking on [Add-In Lines/Points/Polygons] button (For example, you may want to add-in the points that represent your head observations (targets), so that these lie horizontally in the middle of an unstructured grid cell. Likewise, you can conform the grid around geological features such as faults/fractures). An Add-in may be included or excluded in the grid creation, by checking or unchecking the corresponding check box, respectively. When an add-in is “checked” it will also be displayed in the adjacent 2D Viewer preview window.

- Click the [Next>>] button to proceed.

In the second dialog, you can define various discretization settings for the horizontal grid and adjust refinement levels and smoothing around the line and point add-ins.
For now, you will just use the default settings.

- Click the [Generate] button (located in the bottom middle section of the window). A preview of the USG grid will appear in the adjacent 3D viewer. If you are not satisfied with the grid, you can modify the settings and regenerate the grid by selecting the Generate button again.
• Click [Finish] to generate the numerical grid and close the window.

The Unstructured Grid will be added as a new data object to the Conceptual Model tree, under the Simulation Domain folder, and should appear as shown below.
• Click [Next Step] to proceed.

**Generate the Numerical Model**

You should arrive at the 'Convert to MODFLOW-USG Model' workflow step. You are now ready to create a numerical model from the unstructured grid you created, using the property zones and boundary conditions defined in the conceptual model.

• Click the 'Convert to MODFLOW-USG Model' step in the list of workflow steps (if you're not already there)
• Click the [Convert to Numerical Model] button
The conversion process will begin; this may take a few moments. Note that for larger grid sizes and more complex boundary conditions, this conversion process may take several minutes.

After the conversion is complete, a new workflow tab will appear, titled "UnStructuredGrid1-Run1". This workflow will provide you the tools and displays for navigating the MODFLOW-USG Numerical Model (keeping in mind that you were previously navigating and working with a Conceptual Model). More details on the specifics of each step can be found in the section MODFLOW-USG Numerical Modeling Workflow.
In addition, you will now see new items in the Model Explorer, under the UnstructuredGrid1 node (in the lower left corner of your window), with a Run folder containing Inputs (Properties and Boundary conditions) and Outputs (Heads and Drawdown, which will be added after a successful MODFLOW-USG run). For the boundary conditions, you will see nodes for groups of boundary condition cells that were generated: Constant Head boundaries, River, and the Pumping Wells.
Define Properties

The first step that appears in the workflow is Define Modeling Objectives step. This step allows you to define type of model which will be run and to define many default property values. These values should be identical to the values initially defined in the 'Define Modeling Objectives' step during the conceptual modeling workflow. For now we will retain all the default values.

- Click [Next Step] to proceed.

The Define Properties workflow step will appear, which lets you edit Conductivity, Storage, and Initial Heads values in your numerical model.
By default, Conductivity will be displayed in the 3D viewer. All UnStructured grid cells will be drawn and colored by the Kx.

In order to gain a better perspective, you need to apply a Vertical Exaggeration and rotate the 3D Viewer.

- Locate the 'Exaggeration' field in the toolbar above the 3D Viewer
- Type: 40, then press Enter on your keyboard.

Now you will rotate the 3D view to see the property zones from a side-view perspective.

- Left click with the mouse near the bottom middle section of the 3D display
- Hold down the left-mouse button, and drag your mouse upwards towards the top of the VMOD Flex window, then release the mouse button. By clicking and dragging the mouse in the viewer window you can position the image however you like. You may need to select the rotate button from the toolbar on the right side before clicking and dragging the mouse.
- Above the 3D Viewer, you will see a set of standard navigation tools for zoom in/out, pan, and rotate which you can use to further manipulate the view.
In order to change how the parameter values are displayed, you need to load the settings. (This step is optional in the Tutorial exercise) Right-click on 'Conductivity' in the Model Explorer, and select [Settings]. This will allow you to render by the parameter value (eg. Kx) instead of by ZoneID, and also show cross-sectional slices and color maps. Take a moment to experiment with these Settings, and when you are finished, click the [OK] button to close the Settings.

In order to see another parameter group, you need to de-select (remove the checkbox beside) Conductivity in the Model Explorer, then select (add a check-box beside) the new parameter group, eg. Initial Heads. Take a moment to experiment with these options.

- Click [Next Step] (Next Step) to proceed.

Define Boundary Conditions

At this step in the workflow, you can see and edit the numerical (cell) representations of the Boundary Conditions that were generated from the conceptual model objects. The constant heads on the north and south boundaries are shown as red cells, the river is shown as blue cells, and the pumping wells are beige. You display should now appear similar as below:
Take a moment to zoom into the river and constant head cells to see the boundary condition cell geometry. You may need to switch layers to see the boundaries.

Each boundary condition will be colored differently. All wells and line-based boundary conditions will be shown by default.

The display settings for a group of boundary condition cells can be adjusted through the settings. We will turn on the Cell ID for the well cells.

- Locate 'PumpingWell1' in the Model Explorer, under 'Run1/Input/Boundary Conditions/Wells'
- Ensure 'PumpingWell1' is being displayed by activating the checkbox (✓)
- Right click on this item and select [Settings...]
- Expand [Style]>[Cells]
- ✓ beside 'Show Cell ID labels'
- Click [OK]
- You should now see the Cell ID drawn beside the two well cells in the 3d viewer. If these are hard to read, try changing the grid color by editing the 'UnstructuredGrid1' settings. You can also change the background color by right-clicking in the 3D viewer and selecting 'Background Color'. You will need to zoom into the two pumping wells in order to see the Cell ID label. The Cell ID can assist when editing the numerical values, or when cross-checking the values in the .WEL package. For more details on editing the numerical values for boundary conditions, see View and Edit Boundary Conditions

- Click [Next Step] (Next Step) to proceed.

Define Head Observations

We will map the head observation wells (raw data) to the numerical model, so that this data can be used for model calibration. From the 'Select Next Step' workflow window, select the 'Define Observation Wells' option:

- Click the 'Select Next Step' workflow step
- Click the 'Define Observation Wells' button
- Select the 'Head_Observation Wells' data object from the Data Tree in the top-left panel of the window.
- Click on the [ ] button located above the 3D Viewer.
- The observation wells will be added to the display and the Model Explorer. These will appear as two green points, one is located in the top left corner, the other in the bottom right corner. (Note: some display settings have been changed from the default values in the image below; vertical exaggeration set to 0, observation well points increased size, )
Define Zone Budget Zones

It is also possible to define Zone Budget zones for MODFLOW-USG models. To do this, return to the 'Select Next Step' workflow step, and select 'Define Zone Budget Zones'. We will define a zone budget zone for our pumping wells, so that we can estimate the volume of water pumped extracted the course of the simulation. We will also assign a zone budget zone to the cells which contain the river boundary condition (i.e. southern boundary).
Click the 'Select Next Step' workflow step in the workflow navigator
Click the 'Define Zone Budget Zones' button
Under the toolbox, click 'Assign' > 'Using data object...'
From the 'Data Tree' select the 'river' data object
Click on the button in the 'Select geometry object' window
Click OK
The 'Create new zone budget zone' window will appear
Click 'New' to create a new zone budget zone (i.e. Zone 2, blue)
Ensure 'Layer 2' and 'Layer 3' are selected in the 'Assign to layer' frame
Click OK
Now assign the zone budget zone for our pumping wells
Zoom to the location of the pumping wells (lower-right corner)
Under the toolbox, click 'Assign' > 'Single'
Click both cells containing pumping wells (i.e. the two 'central' cells)
Click 'Finish' under the toolbox
Click 'New' to create a new zone budget zone (i.e. Zone 3, green/teal)
Ensure 'Layer 2' and 'Layer 3' are selected in the 'Assign to layer' frame
Click OK
In the Model Explorer, you can activate the 'Zone Budget1' object to display the distribution of zone budget zones. If you turn this object on and zoom to the lower-right corner of the model you should be able to view all three zone budget zones at once (see image below):

Define Particles

You should now see the 'Select Next Step' workflow step. At this stage you can define several optional model elements such as particles (for mod-PATH3DU simulations), Zone Budget Zones (for ZoneBudget-USG simulations), and observation wells (for calibration charts).
Let's define some backward tracking particles around our pumping well objects. These will help us to delineate a capture zone for these wells.

- Click 'Define Particles' in the 'Select Next Step' workflow window
- Under the toolbox, click 'Assign' > 'Using well object...'
- The 'Create New Particles' window will appear
- In the Data Tree select the 'VMOD Imported Wells' data object
- Click on the button where it says 'Select object'
- In the 'Particles along screen' field, specify '3' as the '# of Circles'
- When you're finished, the 'Create New Particles' window should look like the image below:
Click 'OK' to define the particles

You should see a circle of particles appear around the pumping wells in the lower-right corner of the model, as shown below (note: background color changed, and image is zoomed to location of the particles):
Click [Next Step] (Next Step) to proceed to the 'Select Run Type' workflow step.

Click [Next Step] (Next Step) to proceed to the 'Single Run' workflow step.

Activate Zone Budget and particle tracking:

- ✔ ZONEBUDGET-USG
- ✔ Particle Tracking

Click [Next Step] (Next Step) to proceed to the 'Translate' workflow step.

**Translate (Create MODFLOW-USG Input files)**

At the Translate step, you create the packages for MODFLOW-USG, and you can preview and make adjustments to the input files where needed.
Under 'MODFLOW-USG' > 'Settings' > 'Run Type', select 'Transient'.

Click [Translate] to create the MODFLOW-USG packages.

The translation will begin, it should complete in approximately 5-10 seconds. At this stage, if any errors or warnings are encountered with any of the packages, you will be notified. Take a moment to preview the MODFLOW-USG input files; each package will appear as a separate tab across the top of the window.

When you are finished, you can proceed to the run step.

Click [Next Step] to proceed to the 'Run' workflow step.

**Run MODFLOW-USG**

Click the [Run] button to run MODFLOW-USG.

The MODFLOW-USG Engine will start running and show progress of the model run in the main viewer window. After the model converges you can view the summary of the mass balance for the last time step in each stress period.
• Click [ ] (Next Step) to proceed.
• Click the [View Maps] button when presented with the desired results options.

View Maps (Heads)
You will then see cell rendering of calculated heads in the 3D Viewer:
By default the view is plan (from above); take a moment to rotate the view as previously described, and apply the desired vertical exaggeration (40 is ideal for this example). The default option is to the selected layer as a slice. You can display heads along a specific layer, or cross-section by adjusting the settings.

Right-click on 'Heads' from the Model Explorer (under 'Run1/Output/Flow'), and select [Settings] from the pop-up menu:

Next from the 'Settings' tree on the left, expand 'Slice', turn on the check box beside 'Show Slice', and change the Slice Type to 'Row'. You will also change the 'XZ Slice
Position' to '50'. This indicates that a cross-section will be displayed in the X direction, including all cells which intersect the desired location. The XZ slice position value is expressed as a percentage of the entire distance in the selected direction.

- Click the [OK] button to apply these changes to the 3D View, and close the 'Settings' window.
- You should then see cells rendered along the X direction, approximately half-way through the model domain, as shown below:
For Transient models, the 3D Displays can be updated to display calculated heads (or drawdown) from different time steps. Use the time-step picker menu above the 3D viewer to choose from the available time-steps. Additional options for time steps are located in the 'Settings'.

Repeat the same steps above to load the 'Settings' window. Expand 'Style', and select the 'Time' node from the tree, and you should see the following display:
• In the 'Settings' window, activate the 'Show time label' checkbox.

• In the main window, above the 3D View, you should see a 'Time-Step Picker' menu, which lists available output times. Choose the desired output time from this list and the 3D View will update with the calculated heads from that time step. Take a moment to experiment with these options.

• If you select the final time step, the resulting view should look like the following image:
Export
The calculated heads and drawdown can be exported to shapefile (point or polygon) or .CSV, for further post-processing. This option is available by right-clicking on the 'Heads' item on the Model Explorer, and selecting [Export...]. The heads can be exported to either point or polygon shapefiles, with attributes. This option is not covered in this Tutorial exercise.

View Maps (Heads)
Displaying particle pathlines is very easily done. Reset the 3D viewer window to it's original state by clicking the 'Reset View' button, then activate the pathline outputs from the Model Explorer.

- Click the [ ] (Reset View) button in the 3D viewer toolbar
- ✓ Well Group Pathlines under the Model Explorer, to display the particle pathlines
- You should then see the backward tracking particles that we had placed around the pumping wells, as shown below:
View Charts (Calculated vs. Observed)

In this section you will learn how to compare the observed concentration data to the concentration values calculated by the model.

- Select the [View Charts] item from the workflow tree, under 'View Results'
- Select the 'All Obs' check box in the Charts panel, as shown below.
- Click the [Apply] button. You will then be presented with the graph for Calculated vs. Observed Heads.
You can change the chart type to display 'Time Series' graph, and see the corresponding charts for the desired observation points. An example is shown below, with "All Obs." visible.

View Zone Budget Results

To review the results of the ZoneBudget analysis simply click the 'Zone Budget' button located at the top-left corner of the 'View Charts' workflow step:

- Click the 'Zone Budget' button located above the 'Chart Type' menu
- You should see the following window open (the four sub-windows have been rearranged to better display the contents of the Zone Budget window):
Zone budget analyses are an easy way for you to better understand the water budget in areas of interest within your model.

- Once you have reviewed the Zone Budget results you may close the Zone Budget window.

💡 Evaluate Multiple Numerical Models... and find the best one!!

If, after running the numerical model for your project, you find that the grid is not suitable or stable, you can return back to the "Define UnStructured Grid" step, create a new grid (with different refinement levels), and generate a new numerical model. You can then translate and run this model. In this fashion, you can evaluate multiple numerical models, with very little effort, in order to find the best balance between model run times and accuracy.
Defining Unstructured Q-Grids

The conceptual modeling workflow allows the user to test a variety of different grid types very quickly and easily. In this section we will return to the conceptual model to generate a new unstructured grid type (quadtree grid or Q-grid), merge the conceptual model and Q-grid into a single MODFLOW-USG numerical model and re-run the model.

- At the top of the grid view you will see a list of active tabs. Click the 'Conceptual Model 2' tab to return to the conceptual modeling workflow.
- Return to the 'Select Grid Type' workflow step, as shown below

This time we will select the 'Define Unstructured Q-Grid' option
- Click the 'Define Unstructured Q-Grid' button
The 'Create Unstructured Q-Grid' window will appear, as shown below:

The 'Create Unstructured Q-Grid' window includes a list of the data objects which have been used to define model elements (e.g. polyline, polygon or points data objects used to define boundary conditions). Using the 'Refine' and 'Refine to Min' buttons in the table allow you to perform successive refinements around the selected data object. The 'Refine' button will split all cells containing the selected data object into four equally sized cells. The 'Refine to Min' button will perform successive refinements until a given minimum area threshold (user defined, in the 'Min Area' column) is reached. Experiment with these options and perform some refinements around the pumping well, river and constant head boundary objects.
• Click the 'Refine' button in the 'PumpingWell1' row twice (x2)
• Click the 'Refine' button in the 'River 1' row once (x1)
• Type '500' in the 'Min Area (m^2)' field for the 'Polygon1 (Boundary)' data object
• Click the 'Refine to Min' button in the 'Polygon1 (Boundary)' row
• When you're finished the 'Create Unstructured Q-Grid' window should look like the image below (note the refined grid cells around the selected data objects; also note that the 'count' column tracks the number of refinements performed for each data object):

• Click OK

The unstructured Q-grid is now available for the conceptual to numerical model conversion. Follow the same steps as above to convert the conceptual model to a numerical model, review properties and boundary conditions, apply zone budget zones and/or particles, and finally translate and run the unstructured Q-grid model.

Once you have completed these steps you should see results similar to the following images displaying head distributions, particle pathlines and zone budget results, respectively (note: results are displayed for a steady-state simulation):
*****This concludes the 'Groundwater Flow Modeling with MODFLOW-USG' tutorial.*****
4 Working with Your Data

The modeling workflows in Visual MODFLOW Flex are data driven. This means that you first need to load or create the appropriate data objects in order to use these at a particular step in the workflow. See the following table for some common examples:

<table>
<thead>
<tr>
<th>If you have...</th>
<th>First you should...</th>
<th>Then you can...</th>
</tr>
</thead>
<tbody>
<tr>
<td>River locations in a shapefile</td>
<td>Import these as Polyline data objects</td>
<td>Select this data object when creating a river boundary condition</td>
</tr>
<tr>
<td>Geological layers in a Surfer .GRD or ASCII GRD</td>
<td>Import these as Surface data objects</td>
<td>Select these data objects when defining horizons</td>
</tr>
<tr>
<td>Aerial photo with river locations</td>
<td>Import this as a Map image Then Create a new Polyline data object Digitize the polyline in a 2D Viewer</td>
<td>Select this new polyline data object when creating a river boundary condition</td>
</tr>
<tr>
<td>XYZ points for geological contacts</td>
<td>Import these as Points data objects Then &quot;Create Surfaces&quot; from these data objects</td>
<td>Select these surfaces when defining horizons or numerical model layers</td>
</tr>
<tr>
<td>Raster Grid of Kx or Recharge data (from Surfer, ESRI .GRD)</td>
<td>Import these as surface data objects</td>
<td>Select these data objects when defining properties or Recharge boundary conditions</td>
</tr>
<tr>
<td>Visual MODFLOW (Classic) project or USGS MODFLOW files</td>
<td>Select the Numerical model workflow after creating a project</td>
<td>Select these files when prompted to Import Model. Learn more...</td>
</tr>
</tbody>
</table>

💡 Learning more

- See Importing Data for details on importing GIS data, images, Excel spreadsheets, or XYZ text.
4.1 Importing Data

Visual MODFLOW Flex supports importing data from various standard data types to allow you flexibility in constructing your conceptual model. Data can be imported and used in several ways; spatial data can be used to delineate and visualize geometry of structural zones, horizons and features of your conceptual model, while attribute data can be used in assigning properties to structural zones and attributes to boundary conditions.

About the Import Process

The data import process in Visual MODFLOW Flex varies slightly depending on the data type being imported. However, the import process generally consists of the following steps:

1. Select the data type and source file
2. Specify the coordinate system (including projection and datum) of the source data
3. Map the source data fields with required target fields, and map attribute fields
4. Data preview and validation

The following sections provide additional information on the import process for each data type:

- Points
- Polylines
- Polygons
- Surfaces
- Wells
- 3D Gridded Data
- Maps
- Time Schedules
- DXFs

4.1.1 Points

Points represent discrete locations in space (XYZ) where attribute information is known. Examples of points data include: ground surface or subsurface elevations, well tops,
locations with known aquifer hydraulic properties, etc. Typically, this information may come from drilling wells or monitoring events where information is gathered from a specific location.

Once imported, points data can then be interpolated to generate surfaces. These surfaces can be used to create conceptual model horizons, or for defining property values for structural zones. For more information on creating surfaces from points data see Creating Surfaces.

Visual MODFLOW Flex supports the following file types for points data:

- Shapefile (*.SHP)
- AutoCAD (*.DXF)
- Text (*.TXT)
- Comma-Separated Values (*.CSV)
- Access Database (*.MDB)
- Excel (*.XLS)

For Points data, the following data must exist in separate columns, in your source data:

- X
- Y
- Elevation
- Attribute 1 (optional) (e.g., conductivity)
- Attribute 2 (optional) (e.g., layer 2 elevation)
- etc..

To import points data, follow the steps below:

- Right-click in the Data Explorer, and select Import Data... from the pop-up menu. The import dialog will open:
Select Points from the Data Type drop-down list. Click the [...] button and locate the source file.

Enter a Name and a Description (optional) for the imported data, and click [Next] to continue.

- The next step allows you to preview the source data before importing, and will vary depending on which file type is selected.

For .CSV and .TXT files (shown below), select the appropriate delimiter from the Delimiters frame, e.g., if it is a *.CSV file, you would select “comma”. Specify which row to start importing from using the From row selector.
For .XLS files (shown below), select which Excel worksheet to import from the Select Worksheet drop down list. Also, you can choose which row to import from using the From row selector.
For .MDB files (shown below), you can choose to import data from a table or a query, by selecting either the View Tables or View Queries radio button. Select the desired query/table from the Select Table or Query drop down list box.
Next, select the Coordinate System of the data being imported. If the coordinate system is
different than the one defined in the project settings, VMOD Flex will perform a geo-
transformation, converting all coordinates to the project’s coordinate system. Click the
[Next>>] button to continue to the next step.
Next, set your Data Mapping by mapping columns in the source data to the target fields in Visual MODFLOW Flex. A read-only preview of the source data is presented. The process of data mapping is described in the following section in greater detail.
Data Mapping

The first column in the Data Mapping table, named Target_Fields, contains the required target fields for the data object. The second column, named Map_to, allows you to match the fields in the source data to each required target field.

If the column labels in the source data are identical to the labels of the target fields, Visual MODFLOW Flex will automatically map the columns for you. However, if the labels differ, you must map the columns manually.

To map a source field to a target field, select the corresponding source field from the drop list box in the Map_to column. The drop down list displays the column headers in the data source file.
For example, in the figure above, the elevation field in the source data is labeled “Z”. To map this field to the target field “Elevation”, select “Z” from the adjacent drop down list.

If the file you are trying to import is a 3D Shapefile (with Elevation defined), then Visual MODFLOW Flex should automatically detect this, and you should not need to map the Elevation attribute at this step. You can also specify a data field as the source of elevation information, if appropriate.

Source fields that are not required, can be mapped by creating a new attribute. To create a new attribute, click the Add a new attribute button. A new row will be added to the Data Mapping table.

In the Map_to column, select the desired attribute field in the source data, from the combo box. Repeat for additional attributes. You can delete a mapped attribute by selecting the row from the Data Mapping table, and then clicking the Delete button.

Use the Unit Category and Unit columns to define the units of a mapped field. If the specified units are different than those defined in the Project Settings, Visual MODFLOW Flex will automatically convert the data in the source file to the default project units.

The Multiplier column allows you to multiply all values in the mapped field by a specified multiplier value.

The Data Type column allows you to define the data type. Select from the following options: Numeric, Text, Boolean, Date and Time. For example, if the mapped column contains text data, select Text from the drop down list.

Once the data mapping is complete, click the [Next] button to continue to the validation dialog.

### Data Validation

The final step involves validation of the data being imported. This step will ensure that the data set contains valid data for each of the mapped fields.
In the top half of the dialog, Visual MODFLOW Flex will list any mapped fields that contain invalid data, along with a reason for why they are deemed invalid. The data validation rules for each mapped column are as follows:

- X and Y values must be a numeric value, and present in each row of the mapped data.
- Data columns will be deemed invalid if Visual MODFLOW Flex detects a null (blank) field.
- Data in each column must satisfy the specified data type.

If invalid data exists, you can choose to import this data anyway. Otherwise, you can select the Do Not Import Rows with Warnings check box, and Visual MODFLOW Flex will not import any rows deemed invalid.

In the bottom half of the dialog, there are two options:

- Show only errors and warnings: When selected, only the records deemed invalid will be shown in the preview table. Records that are deemed invalid will be colored either red (error) or yellow (warning).
- Show this amount: When selected, you can view a specified number of records in the preview table below. Enter a value, and then click the [Apply] button to show the records (both valid & invalid) in the preview table.

When you import pumping well schedule data or head/concentration observation data in Absolute time format, Visual MODFLOW Flex will check the date values against the start date you defined in the modeling objectives;
For pumping well schedules, any wells that have dates before the model start date will be flagged as warning, since these data cannot be used when you create Pumping Wells Boundary condition object. Likewise
For head/concentration observations, any wells that have dates before the model start date will be flagged as warnings, since these observations cannot be used as calibration points in the numerical model.

Click the [Finish] button to import the data. Once imported, a data object will be added in the Data Explorer.

4.1.2 Polylines
Polyline data consists of a series of points (vertices) connected by lines. Polyline data objects can be used in Visual MODFLOW Flex for defining geometry and assigning attributes to linear boundary conditions, such as River and Drain boundary conditions. Polylines may also be useful to visualizing geographic features such as river and road networks.

Visual MODFLOW Flex supports the following file types for polyline data.
- Shapefile, *.SHP
- AutoCAD, *.DXF

To import polyline data, follow the steps below:
- Right-click in the Data Explorer, and select Import Data... from the pop-up menu.
- Select Polyline from the Data Type drop down list.
- Click the [...] button and locate the source file.
- Enter a Name and a Description and then click [Next] to continue.

The next step involves selecting the coordinate system of the source file, and will vary depending on which file-type is selected for the source file.
For .SHP files, if the associated .PRJ file is located in the same location as the source file, Visual MODFLOW Flex can automatically detect the coordinate system of the source data and will perform a geotransformation if the coordinate system is different than that defined in the project settings. If a .PRJ file is missing, than you will be prompted to select the Coordinate System for the data being imported.
For .DXF files, you will always be prompted to select the Coordinate System of the selected source file.
Click the [Next] button to continue.
If the file type is .SHP, the next step involves creating attributes. If you are importing from .DXF file, you can skip this step.
If the file you are trying to import is a 3D Shapefile (with Elevation defined), then Visual MODFLOW Flex should automatically detect this, and you should not need to map the Elevation attribute at this step. You can also specify an attribute field as the source of elevation information. For example, if you have contour information.

This dialog allows you to import shapefile attributes. To create a new attribute, click the Add a new attribute button. When selected, a new row will be added to the Data Mapping table.

In the Map_to column, select the desired attribute field in the source data, from the combo box. Repeat for additional attributes. You can delete a mapped attribute by selecting the row from the Data Mapping table, and then clicking the Delete button.

For a description of the Unit Category, Unit, Multiplier and Data Type columns, please refer to section "Data Mapping" section.

Click the [Next] button to continue.

The final step involves validation of the data being imported. This step will ensure that the data set contains valid data for each of the mapped columns.

For .SHP files, please refer to "Data Validation" section for more information on the data validation step.

For .DXF files, the following dialog will show, indicating the number of polylines that will be created from the source file.
Click the [Finish] button to complete the polygon importing process. Once imported, a polyline data object will be added to the Data Explorer.

You can visualize your polyline data objects by displaying them in a 2D or 3D Viewer or the Layer View within the Numerical Workflow. For example a DXF of the rivers:
You can adjust the Style settings for the DXF in each of the viewers. For more information on adjusting please review Points, Polylines, and Polygons section.

4.1.3 Polygons

Polygons are closed shapes consisting of vertices, line segments and have at least 3 sides. Polygons can be used in Visual MODFLOW Flex in the following ways:

- To define the horizontal boundary of a conceptual model
- To define the geometry and attributes of horizontal boundary conditions, e.g., recharge, specified-head.
- To define the geometry and attributes of property zones.
- To visualize spatial variation of geographic features using various style settings.
- Visual MODFLOW Flex supports the following file types for polygon data.

The following polygon file types are supported:

- Shapefile, *.SHP
- AutoCAD, *.DXF
To import polygon data, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Polygon from the Data Type drop down list box.
- Click the [...] button and locate the source file.
- Enter a Name and a Description for the imported data, and click [Next] to continue.

The remaining workflow for importing polygon data is very similar to that of importing Polylines. For more information on how to import polygons, please see Importing Polylines

4.1.4 Surfaces

Surface data consists of an ordered array of interpolated values at regularly spaced intervals that represent the spatial distribution of an attribute, e.g., digital elevation models. Surface data can be used in Visual MODFLOW Flex in the following ways:

- To define the horizons (structural zone vertical boundaries) of a conceptual model.
- To define the spatial distribution of a boundary condition attribute.
- To define the spatial distribution of a property zone attribute, e.g., conductivity, initial heads.
- To visualize the spatial variation of model features, e.g., surface topography, water table elevation, etc.

Visual MODFLOW Flex supports the following surface file types:

- SURFER Grid Files (*.GRD)
- ESRI ASCII Raster Files (*.ASC, *.TXT)
- USGS Digital Elevation Model (*.DEM)

To import surface data, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Surface from the Data Type drop down list box.
- Click the [...] button and locate the source file.
- Enter a Name and a Description for the imported data, and click [Next] to continue.

**Please Note:** Surface files that contain a large quantity of data points may require substantial time to import into Visual MODFLOW Flex.

Next, select the Coordinate System of the data being imported. If the coordinate system is different than the one defined in the project settings, Visual MODFLOW Flex will perform a geotransformation, converting all coordinates to the project’s coordinate system. Click the [Next] button to continue to the next step.
Surface data will usually only consist of three columns: X, Y and Attribute Data (elevation, conductivity, etc). Visual MODFLOW Flex will automatically map the source columns to the target fields. You can preview the mapped data before importing into Visual MODFLOW Flex.
Click the [Next] button to finish the import process. Upon importing, a new data object will be added to the Data Explorer.

### 4.1.4.1 SURFER Grid Files

Visual MODFLOW Flex supports the import of SURFER ASCII Grid Files (*.grd).

**Format**

The format of a Surfer ASCII grid file consists of six items:

```
DSAA
Nx Ny
Xmin Xmax
Ymin Ymax
Zmin Zmax
Z1 Z2 Z3 Z4 ... Znx
Znx+1 ...
...
```

---

© 2019 by Waterloo Hydrogeologic
Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DSAA key</td>
<td>first line of the file must include the text &quot;DSAA&quot; which is a text key used to indicate that the file is a SURFER ASCII grid file</td>
</tr>
<tr>
<td>Nx Ny</td>
<td>Nx - the number of grid cells in the x-direction (integer) Ny - the number of grid cells in the y-direction (integer)</td>
</tr>
<tr>
<td>Xmin Xmax</td>
<td>Xmin - the X-coordinate in the left-/western-most cell Xmax - the X-coordinate in the right-/eastern-most cell</td>
</tr>
<tr>
<td>Ymin Ymax</td>
<td>Ymin - the Y-coordinate in the bottom-/southern-most cell Ymax - the Y-coordinate in the top-/northern-most cell</td>
</tr>
<tr>
<td>Zmin Zmax</td>
<td>Zmin - the smallest value in the surface Zmax - the largest value in the surface</td>
</tr>
</tbody>
</table>

Data Block

The data component of the SURFER ASCII grid follows the header information.

- Cell values should be delimited by spaces.
- No carriage returns are necessary at the end of each row in the raster.
- Row 1 of the data is at the bottom (south) side of the raster, row 2 is the data for the next northward data above row 1, and so on.

Example

An example dataset follows:

```
DSAA
10 15
480000 480275
4864800 4865200
321.162 336.095
```
335.007 334.943 334.974 335.267 335.899 336.095
335.553 333.868 332.647 330.317
334.142 333.998 332.027 330.824 329.693
333.377 334.423 334.818 334.729 334.820 335.349 335.765 335.623
329.890 332.759 330.948 333.985 332.567 331.600
330.137 332.228 333.393 333.985 332.567 331.600
331.413 331.859 332.229 332.919 332.253 331.781
330.973 330.326 329.915 329.304
330.267 331.006 331.457 331.975 331.785 331.566
330.869 330.279 329.891 329.355
329.539 329.721 329.755 329.496
328.748 329.396 329.695 329.593 328.536 327.843
326.796 327.963 328.638 329.024
328.164 328.715 328.879 328.422 326.930 325.949
324.327 325.862 326.817 327.945
327.754 328.045 328.072 327.573 326.338 325.279
322.970 323.706 324.460 326.505
327.139 327.053 326.932 326.492 325.839 325.008
322.361 321.909 322.297 325.212
326.452 326.145 325.915 325.448 325.291 324.766
322.293 321.339 321.549 325.050
325.728 325.247 324.908 324.425 324.513 324.136
322.078 321.162 321.638 326.150
324.672 323.889 323.423 322.897 323.037 322.829
321.355 321.259 322.214 327.178
Imported example dataset, as displayed in Visual MODFLOW Flex.

4.1.4.2 ESRI ASCII Raster Files

Visual MODFLOW Flex supports the import of ESRI ASCII Raster Files (*.asc).

Format

The format of an ESRI ASCII raster file consists of six items:

- NCOLS xxx
- NROWS xxx
- XLLCORNER xxx
- YLLCORNER xxx
- CELLSIZE xxx
- NODATA_VALUE xxx
- Z1 Z2 Z3 Z4 ... Znx
- Znx+1 ...
- ...
- ...
Where:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Requirements</th>
</tr>
</thead>
<tbody>
<tr>
<td>NCOLS</td>
<td>Number of cell columns.</td>
<td>Integer greater</td>
</tr>
<tr>
<td></td>
<td></td>
<td>than 0.</td>
</tr>
<tr>
<td>NROWS</td>
<td>Number of cell rows.</td>
<td>Integer greater</td>
</tr>
<tr>
<td></td>
<td></td>
<td>than 0.</td>
</tr>
<tr>
<td>XLLCORNER</td>
<td>X coordinate of the origin (lower left corner</td>
<td>Greater than 0.</td>
</tr>
<tr>
<td></td>
<td>of the cell).</td>
<td></td>
</tr>
<tr>
<td>YLLCORNER</td>
<td>Y coordinate of the origin (lower left corner</td>
<td>Optional, Default</td>
</tr>
<tr>
<td></td>
<td>of the cell).</td>
<td>is -9999.</td>
</tr>
<tr>
<td>CELLSIZE</td>
<td>Cell size.</td>
<td></td>
</tr>
<tr>
<td>NODATA_VALUE</td>
<td>The input values to be NoData in the output</td>
<td>Optional, Default</td>
</tr>
<tr>
<td></td>
<td>raster.</td>
<td>is -9999.</td>
</tr>
<tr>
<td>DATA BLOCK</td>
<td>See description below</td>
<td></td>
</tr>
</tbody>
</table>

**Data Block**

The data component of the ESRI ASCII raster follows the header information.

- Cell values should be delimited by spaces.
- No carriage returns are strictly necessary at the end of each row in the raster. The number of columns in the header determines when a new row begins.
- Row 1 of the data is at the top (north) side of the raster, row 2 is the data for the next southward data below row 1, and so on.

**Example**

An example dataset follows:

```plaintext
NCOLS 10
NROWS 15
XLLCORNER 480000
YLLCORNER 4864775
CELLSIZE 25
```
<table>
<thead>
<tr>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>324.672</td>
</tr>
<tr>
<td>323.889</td>
</tr>
<tr>
<td>323.423</td>
</tr>
<tr>
<td>322.897</td>
</tr>
<tr>
<td>323.037</td>
</tr>
<tr>
<td>322.829</td>
</tr>
<tr>
<td>321.355</td>
</tr>
<tr>
<td>321.162</td>
</tr>
<tr>
<td>321.638</td>
</tr>
<tr>
<td>326.150</td>
</tr>
<tr>
<td>326.452</td>
</tr>
<tr>
<td>326.145</td>
</tr>
<tr>
<td>325.915</td>
</tr>
<tr>
<td>326.425</td>
</tr>
<tr>
<td>324.513</td>
</tr>
<tr>
<td>324.136</td>
</tr>
<tr>
<td>327.178</td>
</tr>
<tr>
<td>327.525</td>
</tr>
<tr>
<td>327.053</td>
</tr>
<tr>
<td>326.932</td>
</tr>
<tr>
<td>326.492</td>
</tr>
<tr>
<td>326.057</td>
</tr>
<tr>
<td>325.839</td>
</tr>
<tr>
<td>325.008</td>
</tr>
<tr>
<td>322.754</td>
</tr>
<tr>
<td>324.602</td>
</tr>
<tr>
<td>328.072</td>
</tr>
<tr>
<td>327.573</td>
</tr>
<tr>
<td>326.368</td>
</tr>
<tr>
<td>325.279</td>
</tr>
<tr>
<td>322.970</td>
</tr>
<tr>
<td>323.706</td>
</tr>
<tr>
<td>324.460</td>
</tr>
<tr>
<td>326.505</td>
</tr>
<tr>
<td>328.164</td>
</tr>
<tr>
<td>328.715</td>
</tr>
<tr>
<td>328.879</td>
</tr>
<tr>
<td>328.422</td>
</tr>
<tr>
<td>326.930</td>
</tr>
<tr>
<td>325.949</td>
</tr>
<tr>
<td>324.327</td>
</tr>
<tr>
<td>325.862</td>
</tr>
<tr>
<td>326.817</td>
</tr>
<tr>
<td>327.945</td>
</tr>
<tr>
<td>328.748</td>
</tr>
<tr>
<td>329.396</td>
</tr>
<tr>
<td>329.695</td>
</tr>
<tr>
<td>329.593</td>
</tr>
<tr>
<td>328.536</td>
</tr>
<tr>
<td>327.843</td>
</tr>
<tr>
<td>326.796</td>
</tr>
<tr>
<td>327.963</td>
</tr>
<tr>
<td>328.638</td>
</tr>
<tr>
<td>329.024</td>
</tr>
<tr>
<td>329.510</td>
</tr>
<tr>
<td>330.272</td>
</tr>
<tr>
<td>330.688</td>
</tr>
<tr>
<td>330.975</td>
</tr>
<tr>
<td>330.612</td>
</tr>
<tr>
<td>330.271</td>
</tr>
<tr>
<td>329.539</td>
</tr>
<tr>
<td>329.721</td>
</tr>
<tr>
<td>329.755</td>
</tr>
<tr>
<td>329.496</td>
</tr>
<tr>
<td>330.267</td>
</tr>
<tr>
<td>331.006</td>
</tr>
<tr>
<td>331.457</td>
</tr>
<tr>
<td>331.975</td>
</tr>
<tr>
<td>331.785</td>
</tr>
<tr>
<td>331.566</td>
</tr>
<tr>
<td>330.869</td>
</tr>
<tr>
<td>330.279</td>
</tr>
<tr>
<td>329.891</td>
</tr>
<tr>
<td>329.355</td>
</tr>
<tr>
<td>331.413</td>
</tr>
<tr>
<td>331.859</td>
</tr>
<tr>
<td>332.229</td>
</tr>
<tr>
<td>332.919</td>
</tr>
<tr>
<td>332.253</td>
</tr>
<tr>
<td>331.781</td>
</tr>
<tr>
<td>330.973</td>
</tr>
<tr>
<td>330.326</td>
</tr>
<tr>
<td>329.915</td>
</tr>
<tr>
<td>329.304</td>
</tr>
<tr>
<td>333.094</td>
</tr>
<tr>
<td>333.228</td>
</tr>
<tr>
<td>333.393</td>
</tr>
<tr>
<td>333.985</td>
</tr>
<tr>
<td>332.567</td>
</tr>
<tr>
<td>331.600</td>
</tr>
<tr>
<td>330.137</td>
</tr>
<tr>
<td>329.879</td>
</tr>
<tr>
<td>329.704</td>
</tr>
<tr>
<td>329.278</td>
</tr>
<tr>
<td>334.337</td>
</tr>
<tr>
<td>334.291</td>
</tr>
<tr>
<td>334.323</td>
</tr>
<tr>
<td>334.729</td>
</tr>
<tr>
<td>333.095</td>
</tr>
<tr>
<td>331.948</td>
</tr>
<tr>
<td>329.890</td>
</tr>
<tr>
<td>329.682</td>
</tr>
<tr>
<td>329.606</td>
</tr>
<tr>
<td>329.314</td>
</tr>
<tr>
<td>334.743</td>
</tr>
<tr>
<td>334.640</td>
</tr>
<tr>
<td>334.695</td>
</tr>
<tr>
<td>335.157</td>
</tr>
<tr>
<td>334.620</td>
</tr>
<tr>
<td>334.012</td>
</tr>
<tr>
<td>332.142</td>
</tr>
<tr>
<td>330.947</td>
</tr>
<tr>
<td>330.267</td>
</tr>
<tr>
<td>329.528</td>
</tr>
<tr>
<td>334.818</td>
</tr>
<tr>
<td>334.729</td>
</tr>
<tr>
<td>334.820</td>
</tr>
<tr>
<td>335.349</td>
</tr>
<tr>
<td>335.765</td>
</tr>
<tr>
<td>335.623</td>
</tr>
<tr>
<td>333.998</td>
</tr>
<tr>
<td>332.027</td>
</tr>
<tr>
<td>330.824</td>
</tr>
<tr>
<td>329.693</td>
</tr>
<tr>
<td>335.007</td>
</tr>
<tr>
<td>334.943</td>
</tr>
<tr>
<td>334.974</td>
</tr>
<tr>
<td>335.267</td>
</tr>
<tr>
<td>335.899</td>
</tr>
<tr>
<td>336.095</td>
</tr>
<tr>
<td>335.553</td>
</tr>
<tr>
<td>333.868</td>
</tr>
<tr>
<td>332.647</td>
</tr>
<tr>
<td>330.317</td>
</tr>
</tbody>
</table>
4.1.5 Wells

The following types of well data can be imported into Visual MODFLOW Flex:

- Well Head Locations
- Well Head Locations and Screen(s)
- Wells with Head/Concentration Observations
- Wells with Pumping Schedules

Visual MODFLOW Flex supports the following file types for importing well data:

- Excel (*.XLS, *.XLSX)
- Comma-Separated Values (*.CSV)

Import Options

To import wells, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Wells from the Data Type drop down list.
- Click the [...] button and locate the source file.
Enter a Name and a Description for the imported data, and click [Next] to continue.

The next step allows you to preview the source data before importing, and choose a subset of the source data, e.g., a worksheet of an excel file. Select which Excel worksheet to import from the Select Worksheet drop-down list. Also, you can choose which row to import from using the From Row text field.

Visual MODFLOW Flex provides you with various options for importing wells, and associated well data. For example, you can choose to import well heads (Well ID, X, Y, Elevation, Bottom), or you can choose to import wells heads along with related screen locations, pumping schedules, or observation points.

In the Select the type of wells to import frame, choose between Vertical and Deviated (Horizontal).

Next, specify how the vertical data is expressed in the source file. If the data is expressed with respect to a vertical reference datum, e.g., above mean sea level, then select Elevation. If the vertical data is expressed as a depth, e.g., distance from ground to the bottom of the well, then select Measured Depth.

**Time Options:** Time-varying data can be imported in Absolute (eg. calendar date 5/10/2015) or in Relative times (1 day, 30 days, 365 days, etc) where the value is relative to the start time defined in the Modeling Objectives.)
Select the Well Heads Only option to import just the well heads, i.e., X-Y coordinates, elevation, and well depth for each well. Select the Well Heads with the Following Data option to import additional data for each well. Options include:

**Screen ID Location**: Import the screen locations for each well. Select the Pumping Schedule check box to also import related pumping schedules for each screen. This data could be used later to define pumping well boundary conditions.

**Observation Points**: Import observation points for each well. Select either Observed heads, Observed Concentrations, or both.

**Well tops**: Import the elevation (or measured depth) of points along the well path, where geologic formation (contact points) intersect with the well. This data could be used later to generate surface and horizon layers.

⚠️ **Please Note**: For data requirements for each option please see the next step “Data Mapping”

Once you have selected which well data to import, click the [Next] button to proceed to the data mapping.

---

**Data Mapping**

This step requires you to map the columns in the source data to the required target fields. The required fields will vary depending on the type of well data you selected in the previous step. The following sections describe the data mapping for each data type option:

**Well Heads Only**

For importing Well Heads only, you must map the following columns from the source data to the required target fields:

- **Well ID**: a unique value in the source data. If not, any rows containing duplicate Well IDs will be skipped and not imported.
- **X**: the x-coordinate of the well
- **Y**: the y-coordinate of the well
- **Elevation**: the elevation/depth of the well head, and
- **Bottom**: the bottom elevation/depth of the well borehole
Wells Heads with Screens

If this option is selected, you must first map the well heads under the Well Heads tab. Next, click on the Screens tab, and map the appropriate columns from the source data to the following target fields:

- **Screen ID**: unique ID for the well screen
- **Screen top Z**: elevation/depth of top of screen
- **Screen bottom Z**: elevation/depth of bottom of screen
For each well in the source data, the Screen ID must be unique. Also, screens should not overlap within a single well. These requirements will be validated in the final step of the well import process.

If you selected the pumping schedule check box in the previous step, click the Pumping Schedule tab and then map the appropriate columns from the source field to the following target fields:

- **Pumping Start Date**: in MM/DD/YYYY HH:MM:SS format or relative time as a numeric value in the chosen units.
- **Pumping End Date**: in MM/DD/YYYY HH:MM:SS format or relative time as a numeric value in the chosen units.
- **Pumping Rate**: the pumping rate for the current scheduled interval
Please Note: If time is not included in the source data (just the date), Visual MODFLOW Flex will automatically set the time to 12:00:00 am.

Well Heads with Observation Points

When this option is selected, you must first map the well heads under the Well Heads tab (described above). Next, click the Observation Points tab, and map the appropriate columns from the source data to the following target fields:

For Observed Heads:

- Obs Point ID
- Obs Point Z (Elevation/depth)
- Observed Head
- Head Observation Date

For Observed Concentrations:

- Obs Point ID
- Obs Point Z (elevation/depth)
- Chemical
- Observed Concentration
- Concentration Observation Date
The Obs. Point Z (Elevation) will determine in which model layer (cell) the observed head/concentration was measured; at the time when you view charts, VM Flex will find the calculated head value from the same cell, and then use this for comparing your observed value to what MODFLOW calculated. The same is true for Transport (MT3D) runs, but in this case it will take concentration from that cell, and compare to what you observed in the field.

**Well Heads with Well Tops**

When this option is selected, you must first map the well heads under the Well Heads tab (described above). Next, click on the Tops tab and map the appropriate columns from the source data to the following target fields:

- **Top Z**: elevation (or measured depth) of formation
- **Top ID**: formation name, e.g., Sand1, Sand2, Clay etc.
Please Note: the well top data in the source file must be formatted as follows:

<table>
<thead>
<tr>
<th>well ID</th>
<th>Top ID</th>
<th>Top Z (elevation-based)</th>
<th>Top Z (depth-based)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Well1</td>
<td>Fill</td>
<td>52</td>
<td>0</td>
</tr>
<tr>
<td>Well1</td>
<td>Sand1</td>
<td>26</td>
<td>26</td>
</tr>
<tr>
<td>Well1</td>
<td>Clay</td>
<td>6.5</td>
<td>45.5</td>
</tr>
<tr>
<td>Well2</td>
<td>Fill</td>
<td>94</td>
<td>0</td>
</tr>
<tr>
<td>Well2</td>
<td>Sand1</td>
<td>17</td>
<td>77</td>
</tr>
<tr>
<td>Well2</td>
<td>Clay</td>
<td>4</td>
<td>90</td>
</tr>
</tbody>
</table>

Validation Step

The final step in the Well import process is data validation. Visual MODFLOW Flex will validate the mapped data, and highlights any rows that contain invalid data, e.g., null values, wrong assigned data type, duplicate rows etc.
4.1.6 3D Gridded Data

3D Gridded Data refers to 3D grids with attributes assigned to each grid cell. 3D Gridded data can be used in VMOD Flex to visualize heads generated from a MODFLOW run in Visual MODFLOW Flex, or for assigning spatially-variable attributes to boundary conditions and property zones. Visual MODFLOW Flex supports the following file types for 3D Gridded data:

- MODFLOW Heads file, *.HDS
- TecPlot 3D Point Grid File, *.DAT

Please Note: In order to import data from MODFLOW .HDS files, the source file must exist in the folder that contains all associated MODFLOW data files, e.g., .DIS, .NAM etc.

To import 3D Gridded data, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.

Please see "Data Validation" section for more information on data validation.
Select 3D Gridded Data from the Data Type drop down list.
Click the [...] button and locate the source file.
Enter a Name and a Description for the imported data, and click [Next] to continue.
Next, select the Coordinate System of the 3D Gridded data. If the coordinate system is different than the one defined in the project settings, VMOD Flex will perform a geotransformation, converting all coordinates to the project’s coordinate system. Click the [Next] button to continue to the next step.

The next step allows you to specify how the gridded data is to be imported.

When importing a .HDS file, the Gridded data for existing grid option will be automatically selected. The grid dimensions in the source file must be identical to the dimensions of a grid in your project. Select the existing numerical grid from Conceptual Model tree, and then click the button.

When importing a .DAT file, the Gridded data for existing grid option will not be available. When this option is selected, the Grid Origin frame will display. If the gridded data is in model coordinates, specify the Grid Origin and the degree of Rotation. If the grid is in world coordinates, you may leave the grid origin as is.
For both file types, you can specify the grid resolution. To import the full grid dimensions, select the Import the true grid dimensions option.

ℹ️ **Please Note:** Depending on the performance capabilities of your computer, 3D Gridded data containing large volumes of data may take a significant time to import.

To improve importing and viewing performance, select the Import a reduced grid size option, and specify a value in the Import every nth node box. For example, if a value of 2 is defined, then VMOD Flex will only import every other node in the 3D grid.

Next, select the appropriate Data Category, Unit, and Data Type for each of the mapped attributes.
The final dialog in the import process for 3D Gridded shows the grid dimensions of the source data.
Working with Your Data

The Source Dimensions frame displays the Number of Rows, Number of Layers, Number of Columns and Number of Time Steps in the source data.

Finally, click the [Finish] button to import the 3D Gridded data.

### 4.1.6.1 TecPlot 3D Point Grid File

Visual MODFLOW Flex supports the import of 3D-gridded data in the TecPlot ASCII IJK-Ordered data point format (*.dat). The format consists of four items:

1. **TITLE** - used to specify a brief description of the dataset, must be enclosed in quotes
2. **VARIABLES** - used to specify the data variables encoded in the IJK-Ordered input, values must be enclosed in quotes and separated by commas and the values must start with "X", "Y", "Z", followed by any number of spatially distributed data variables
3. **ZONE** - used to specify the indices of the IJK data and the format:
   - I is the number of columns along the X-direction
   - J is the number of rows along the Y-direction
   - K is the number of columns along the Z-direction
   - F is the data format. Note that only the POINT format is supported by Flex
4. **Data block** - includes the coordinates and data values. Each row must contain single-space-delimited values corresponding to the variables specified in Item 2 above. Rows must be in ordered first by layer (K), then by row (J), and finally by column (I). A FORTRAN routine to write the data would look something like:

   ```fortran
   DO k = 1 to nlay
       DO j = 1 to ncol
           DO i = 1 to nrow
               WRITE(*,*) X(i), Y(j), Z(i,j,k), (COND(i,j,k,l), l=1,nvar)
           END DO
       END DO
   END DO
   END DO
   ```

An example dataset follows:

```
TITLE = "Simple 3D-Volume Conductivity Data"
VARIABLES = "X", "Y", "Z", "Kx", "Ky", "Kz"
ZONE I=3, J=4, K=3, F=POINT
   1.0 2.0 1.1 2.21 2.21 0.221
   2.0 2.1 1.2 5.05 5.05 0.505
   3.0 2.2 1.1 7.16 7.16 0.716
   1.0 3.0 1.2 3.66 3.66 0.366
   2.1 3.2 1.5 10.1 10.1 0.101
```
<table>
<thead>
<tr>
<th>3.1</th>
<th>3.4</th>
<th>1.3</th>
<th>13.8</th>
<th>13.8</th>
<th>1.380</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>4.0</td>
<td>1.3</td>
<td>5.21</td>
<td>5.21</td>
<td>0.521</td>
</tr>
<tr>
<td>2.2</td>
<td>4.4</td>
<td>1.4</td>
<td>13.4</td>
<td>13.4</td>
<td>1.340</td>
</tr>
<tr>
<td>3.2</td>
<td>4.8</td>
<td>1.2</td>
<td>18.3</td>
<td>18.3</td>
<td>1.830</td>
</tr>
<tr>
<td>1.0</td>
<td>5.1</td>
<td>1.0</td>
<td>5.13</td>
<td>5.13</td>
<td>0.513</td>
</tr>
<tr>
<td>2.3</td>
<td>5.5</td>
<td>1.1</td>
<td>13.8</td>
<td>13.8</td>
<td>1.380</td>
</tr>
<tr>
<td>3.3</td>
<td>5.9</td>
<td>1.0</td>
<td>19.4</td>
<td>19.4</td>
<td>1.940</td>
</tr>
<tr>
<td>1.1</td>
<td>2.0</td>
<td>2.0</td>
<td>4.43</td>
<td>4.43</td>
<td>0.443</td>
</tr>
<tr>
<td>2.2</td>
<td>2.0</td>
<td>2.2</td>
<td>9.73</td>
<td>9.73</td>
<td>0.973</td>
</tr>
<tr>
<td>3.3</td>
<td>2.1</td>
<td>2.0</td>
<td>14.0</td>
<td>14.0</td>
<td>1.400</td>
</tr>
<tr>
<td>1.1</td>
<td>3.0</td>
<td>2.2</td>
<td>7.27</td>
<td>7.27</td>
<td>0.727</td>
</tr>
<tr>
<td>2.3</td>
<td>3.1</td>
<td>2.7</td>
<td>19.0</td>
<td>19.0</td>
<td>1.900</td>
</tr>
<tr>
<td>3.4</td>
<td>3.2</td>
<td>2.3</td>
<td>25.1</td>
<td>25.1</td>
<td>2.510</td>
</tr>
<tr>
<td>1.1</td>
<td>4.0</td>
<td>2.3</td>
<td>9.98</td>
<td>9.98</td>
<td>0.998</td>
</tr>
<tr>
<td>2.4</td>
<td>4.3</td>
<td>2.5</td>
<td>25.7</td>
<td>25.7</td>
<td>2.570</td>
</tr>
<tr>
<td>3.6</td>
<td>4.6</td>
<td>2.2</td>
<td>36.3</td>
<td>36.3</td>
<td>3.630</td>
</tr>
<tr>
<td>1.2</td>
<td>5.0</td>
<td>2.0</td>
<td>12.2</td>
<td>12.2</td>
<td>1.220</td>
</tr>
<tr>
<td>2.7</td>
<td>5.3</td>
<td>2.3</td>
<td>32.0</td>
<td>32.0</td>
<td>3.200</td>
</tr>
<tr>
<td>3.7</td>
<td>5.5</td>
<td>2.0</td>
<td>40.4</td>
<td>40.4</td>
<td>4.040</td>
</tr>
<tr>
<td>1.1</td>
<td>2.0</td>
<td>3.1</td>
<td>6.68</td>
<td>6.68</td>
<td>0.668</td>
</tr>
<tr>
<td>2.3</td>
<td>2.0</td>
<td>3.3</td>
<td>15.5</td>
<td>15.5</td>
<td>1.550</td>
</tr>
<tr>
<td>3.4</td>
<td>2.0</td>
<td>3.1</td>
<td>21.1</td>
<td>21.1</td>
<td>2.110</td>
</tr>
<tr>
<td>1.2</td>
<td>3.0</td>
<td>3.3</td>
<td>11.8</td>
<td>11.8</td>
<td>1.180</td>
</tr>
<tr>
<td>2.3</td>
<td>3.0</td>
<td>3.8</td>
<td>26.5</td>
<td>26.5</td>
<td>2.650</td>
</tr>
<tr>
<td>3.5</td>
<td>3.2</td>
<td>3.5</td>
<td>39.1</td>
<td>39.1</td>
<td>3.910</td>
</tr>
<tr>
<td>1.2</td>
<td>4.0</td>
<td>3.4</td>
<td>16.9</td>
<td>16.9</td>
<td>1.690</td>
</tr>
<tr>
<td>2.5</td>
<td>4.3</td>
<td>3.9</td>
<td>45.1</td>
<td>45.1</td>
<td>4.510</td>
</tr>
<tr>
<td>3.8</td>
<td>4.5</td>
<td>3.4</td>
<td>58.1</td>
<td>58.1</td>
<td>5.810</td>
</tr>
<tr>
<td>1.3</td>
<td>5.0</td>
<td>3.2</td>
<td>17.8</td>
<td>17.8</td>
<td>1.780</td>
</tr>
<tr>
<td>2.8</td>
<td>5.2</td>
<td>3.5</td>
<td>49.2</td>
<td>49.2</td>
<td>4.920</td>
</tr>
<tr>
<td>3.9</td>
<td>5.4</td>
<td>3.2</td>
<td>65.9</td>
<td>65.9</td>
<td>6.590</td>
</tr>
</tbody>
</table>
4.1.7 HGA Cross-Sections

Visual MODFLOW Flex is capable of importing 3D cross-sections generated by Hydro GeoAnalyst (HGA) data management software.

For information on how to create 3D cross-section in HGA, please refer to the HGA User’s Manual. For HGA product information, please visit the HGA product website: www.waterloohydrogeologic.com/hydro-geoanalyst/ or contact your sales representative or local distributor.
When a cross section is created in HGA's 3D Explorer, a file (*.3XS) is saved in the v3D folder, located in the HGA project folder. By default, the location of this folder is:

C:\Program Files\HGAnalyst\Projects\[Project Name]\v3D

A *.3XS file contains information on the wells and layers of each cross section. The cross section can contain geology, hydrogeology, and model layer interpretation layers. When imported into VMOD Flex, this data can be used for generating surfaces and horizons from interpretation layers, or simply for visualization purposes.

To import a cross section file in VMOD Flex, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Cross Section from the Data Type drop down list.
- Click the [...] button and locate the source file.
- Enter a Name and a Description for the data, and click [Next] to continue.
- Next, select the Coordinate System of the cross section. If the coordinate system is different than the one defined in the project settings, VMOD Flex will perform a geotransformation, converting all coordinates to the project’s coordinate system. Click the [Next] button to continue to the next step.
- The final step involves selecting the elevation units and previewing the cross section data.
At the top of the dialog, specify the cross section elevation units from the drop-down list box. You can choose from metres or feet.

The Source File Data frame contains information about the selected source file. Here you can preview the Number of Cross sections in the source file, along with the cross section names. The Number of wells in the source data is shown, along with the well names. Finally, the number and type of interpretations in the source data are shown, e.g., Model, Geology and HydroGeology.

Click the [Finish] button to import the cross section data. Upon importing, a new cross section data object will be added to the Data Explorer.

### 4.1.8 Maps

Site maps of the model region, such as aerial photographs, topographic maps and satellite imagery, are often useful for gaining a perspective of the dimensions of the model, and for locating important characteristics of the model. Although maps do not contain any specific data used in the calculations, and the presence of a map does not influence the results of the simulation, they are useful for enhancing visualization of the model.

Visual MODFLOW Flex supports the following raster graphics file types:

- Bitmap (*.BMP)
- Tagged Image Format (*.TIF)
• JPEG Interchange Format (*.JPG)

**Please Note:** When a raster image is imported into Visual MODFLOW Flex, the source file is copied and saved in the project’s data repository folder. As such, the original file may be modified, moved or deleted without affecting the imported raster image.

To import a map into Visual MODFLOW Flex, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Map from the Data Type drop down list.
- Click the [...] button and locate the source file.
- Enter a Name and a Description for the data, and click [Next] to continue
- Next, select the Coordinate System of the image file. If the coordinate system is different than the one defined in the project settings, Visual MODFLOW Flex will perform a geotransformation, converting all coordinates to the project’s coordinate system.
- Click the [Next] button to continue to the next step.

All raster images must be georeferenced before importing into Visual MODFLOW Flex. If the selected raster image has already been georeferenced, it will have an associated "world" file, and as such, it will not need to be georeferenced in Visual MODFLOW Flex.

**Please Note:** the world file must have the same filename as the image, and must be located in the same folder as the selected source file, in order for Visual MODFLOW Flex to recognize it.

The following table summarizes the supported graphics file types, and the corresponding georeferencing tag files:

<table>
<thead>
<tr>
<th>Raster Source</th>
<th>Corresponding World File</th>
</tr>
</thead>
<tbody>
<tr>
<td>*.BMP</td>
<td>*.BPW</td>
</tr>
<tr>
<td>*.TIF</td>
<td>*.TFW</td>
</tr>
<tr>
<td>*.JPG</td>
<td>*.JPW</td>
</tr>
</tbody>
</table>

If your source image has a world file, you can skip to the final step. If the source file does not have a world file, you will be prompted to manually georeference the raster image. This procedure is described in the following section.

**Georeferencing Images**

Georeferencing a graphics file involves mapping a coordinate system to the individual pixels of the image. When this is required, the following window will display, when importing a raster image:
Toolbar Buttons

The buttons in the top toolbar are described below:

- **Zoom In**: Click-and-drag the mouse to select the zoom area.
- **Zoom Out**: Zoom out of the image.
- **Full Extent**: Zoom completely out so that the entire image is shown.
- **Pan**: When zoomed in, move the image left, right, up or down.
- **Add (Control Point)**: Add a georeference point. See "Adding Georeference Points".
- **Delete (Control Point)**: Delete a selected georeference point.
- **Transform Image**: Assigns coordinates to image pixels using the specified control points.

**Output Region**: This button allows you to save a specified area of the image to the georeferenced file, after the image has been transformed. By default, the output region is the whole image. Click-and-drag a rectangular box on the image to define a new output region, and then click the Save button to save the output region to the georeference file.
Configure Georeferencing Options: Opens the configure georeferencing options dialog box. For more information on these options, please see "Configure Georeferencing Options" section.

Magnification Selector: Select a magnification level from the combo box.

Control Points Table

The control points table contains the specified control points. You can edit an existing control point by selecting the point from the table, and then clicking the Edit button. A dialog box will display prompting you to modify the control point coordinates.

You can also delete a control point from the control points table. To do so, select an existing control point from the grid, and then click the Delete button.

Georeference Information Table

The georeference information table displays information about the georeferenced image including the original image file name and path, the original image size, and coordinate type (projected, local or geographic). It also displays information on the transformation such as the degree of rotation, scale X-Y shift, and the output file name path and image size.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Image File</td>
<td>Q:\VM5\6-Testin</td>
</tr>
<tr>
<td>Image Size</td>
<td></td>
</tr>
<tr>
<td>Width</td>
<td>1453</td>
</tr>
<tr>
<td>Height</td>
<td>1133</td>
</tr>
<tr>
<td>Coord Type</td>
<td>Projected</td>
</tr>
<tr>
<td>Transformation</td>
<td></td>
</tr>
<tr>
<td>Rotation</td>
<td>-0.07</td>
</tr>
<tr>
<td>Scale</td>
<td>1.036</td>
</tr>
<tr>
<td>X Shift</td>
<td>574,140.98</td>
</tr>
<tr>
<td>Y Shift</td>
<td>4,852,789.96</td>
</tr>
<tr>
<td>Output File</td>
<td>Q:\VM5\6-Testin</td>
</tr>
<tr>
<td>Output Region</td>
<td></td>
</tr>
<tr>
<td>Left</td>
<td>574,196.01</td>
</tr>
<tr>
<td>Top</td>
<td>4,863,897.78</td>
</tr>
<tr>
<td>Width</td>
<td>127.27</td>
</tr>
<tr>
<td>Height</td>
<td>74.07</td>
</tr>
<tr>
<td>Output Size</td>
<td></td>
</tr>
<tr>
<td>Width</td>
<td>127</td>
</tr>
<tr>
<td>Height</td>
<td>75</td>
</tr>
</tbody>
</table>
Adding Georeference Points

In order to map pixels of the image to a coordinate system, the image must have at least two georeference points with known world coordinates. To set a georeference point:

- From the top toolbar, click on the Add button.
- Click on a map location where the world coordinates are known.
- A georeference point window will appear prompting for the X and Y world coordinates of the selected location.
- Enter the X and Y coordinates for this point.
- Repeat this procedure for additional georeference points.
- When you set a georeference point, it is added to the Control Points Table.

You can improve the accuracy of the georeferencing by adding more than two control points to the image. When the image is transformed, the Preview tab will display the original control points and the corresponding georeferenced points, thus allowing you to visualize the accuracy of the georeferencing.

Once you have set at least two georeference points, click the Transform button to georeference the image. The georeferenced image will then be displayed in the Preview tab.

Editing Georeference Points

To edit a georeference point:

- Select the georeference point from the Control Points Table
- Once selected, click the Edit button (located just beneath the control points table)
- A Georeference point window will appear prompting for the X and Y world coordinates of the selected location.
- Enter the new X and/or Y coordinates for this point.
- Click the [Ok] button.

⚠️ Please Note: You must click the Transform button again in order for the georeferencing to update to reflect the modified X-Y values.

Deleting Georeference Points

To delete a georeference point:

- Select the georeference point from the Control Points Table.
- Once selected, click the Delete button (located just beneath the control points table)

Configure Georeferencing Options

When the Configure Georeferencing Options button is selected, the following dialog will open:
Symbols Tab
This tab allows you to change the style settings of the original control points and the georeference points. Choose a Style, symbol Size and Color. A preview of the symbol settings is shown in the boxes below.

Georeference Image Tab
This tab allows you to define settings for the georeferenced image. Each setting is described below.
When a georeferenced image is rotated, you can fill the areas of empty space with a specified color. Otherwise, leave the check box unchecked and the empty space will show transparent.

Click the color box beside Fill color for empty area to select the color to fill the empty spaces (only if Show Fill Color option is selected).

Use the slider to set the JPEG Image Quality. When set closer to L (low), more compression is used in the saved georeference image file, resulting in a smaller file and poorer quality. When set close to H (high), less compression is used in the saved georeferenced image file, resulting in a larger file with better quality.

From the Interpolation Mode combo box, specify the interpolation method (algorithm) to use when the image is scaled or rotated. The various interpolation methods are briefly described below. Keep in mind, lower-quality interpolation mode will result in a smaller output file, whereas high-quality interpolation modes will result in a larger output file.

- **Bicubic**: Bicubic interpolation. No prefiltering is done. This mode is not suitable for shrinking an image below 25 percent of its original size.
- **Bilinear**: Bilinear interpolation. No prefiltering is done. This mode is not suitable for shrinking an image below 50 percent of its original size.
- **Default**: Default interpolation mode
- **High**: a high-quality mode.
- **HighQualityBicubic**: High-quality, bicubic interpolation. Prefiltering is performed to ensure high-quality shrinking. This mode produces the highest quality transformed images.
- **HighQualityBilinear**: Specifies high-quality, bilinear interpolation. Prefiltering is performed to ensure high-quality shrinking.
- **Invalid**:  
- **Low**: a low-quality mode.
- **NearestNeighbor**: Nearest-neighbor interpolation.

The final step involves previewing the raster image and viewing coordinate information, before importing into Visual MODFLOW Flex.

The Map Coordinates frame provides the georeferenced coordinates of the Top Right and Bottom Left corners of the image. The path of the georeferenced image, and the associated georeference tag file is also shown.

Click the [Finish] button to import the map into Visual MODFLOW Flex.
4.1.9 Time Schedules

Time schedule data generally contains time data for one or more attributes. It can be used in Visual MODFLOW Flex to define the stress periods for transient boundary condition attributes, e.g., recharge, river stage etc. The following file types are supported for time schedule data:

- Excel (*.XLS)

Time schedule data can be imported using either an absolute or relative time format. An example of an absolute time schedule is shown below:

<table>
<thead>
<tr>
<th>Time</th>
<th>River Stage</th>
</tr>
</thead>
<tbody>
<tr>
<td>11/01/2008</td>
<td>16.18</td>
</tr>
<tr>
<td>11/15/2008</td>
<td>16.01</td>
</tr>
<tr>
<td>12/01/2008</td>
<td>16.12</td>
</tr>
<tr>
<td>12/15/2008</td>
<td>16.29</td>
</tr>
<tr>
<td>etc...</td>
<td></td>
</tr>
</tbody>
</table>

An example of a relative time schedule is shown below:

Starting Date: 11/01/2008

<table>
<thead>
<tr>
<th>Time</th>
<th>River Stage</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>16.18</td>
</tr>
<tr>
<td>15</td>
<td>16.01</td>
</tr>
<tr>
<td>30</td>
<td>16.12</td>
</tr>
<tr>
<td>45</td>
<td>16.29</td>
</tr>
<tr>
<td>etc...</td>
<td></td>
</tr>
</tbody>
</table>

To import time schedule data, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Time Schedule from the Data Type drop down list.
- Click the [...] button and locate the source file.
- Enter a Name and a Description for the imported data, and click [Next] to continue.
- The next step allows you to preview the source data before importing, and choose a subset of the source data, e.g., a worksheet of an excel file. Select which Excel worksheet to import from the Select Worksheet drop down list. Also, you can choose which row to import from using the From Row text field.
- The next step involves selecting the type of time data used in the source file.
If the time values are expressed in relative time select the Relative option, and specify the starting date and time from the combo boxes. If the time values are expressed in absolute time, select the Absolute option.

Click the [Next] button to continue.

The next step involves data mapping and creating attributes. The required target fields will vary depending on which option was selected in the previous step, e.g., Absolute or Relative.
If you selected Absolute, the required target field will be Start Date and Time, and if you selected Relative, the required target field will be Relative Time. Map the time field in your source data to the required target field.

Next, create a new attribute, and map any associated parameter, e.g., recharge, in the source data to the new attribute. If necessary, repeat for additional parameters in the time schedule.

Data mapping and creating new attributes are described in the "Data Mapping" section.

The final step in the time schedule import process is data validation. Visual MODFLOW Flex will validate the mapped data, and highlights any rows that contain invalid data, e.g., null values, wrong assigned data type, duplicate rows etc.

Please see "Data Validation" for more information on data validation.

Click the [Next] button to import the data. Once imported, a time schedule data object will be added in the Data Explorer.
4.1.10 DXFs

Site plans and other spatial information can be useful for gaining a perspective of the dimensions of the model, and for locating important spatial features within the model domain. Although these maps may not contain any specific data used in the calculations, and the presence of a map does not influence the results of the simulation, they are useful for enhancing visualization of the model and in the decision making-process (e.g. what are the simulated results within a given property line?).

Visual MODFLOW Flex supports import and visualization of AutoCAD drawing exchange files (DXFs) for use as a vector basemap.

⚠️ Please Note: All entities within the DXF file will be processed for import and any compatible entities will be imported into the project and displayed using the same color. As such, it may be useful to remove entities that are not relevant to the model prior to import.

To import a DXF into VMOD Flex, follow the steps below:

- Right-click in the Data Explorer and select Import Data... from the pop-up menu.
- Select Dxf from the Data Type drop down list.
- Click the [...] button and locate the source file.
- Enter a Name and a Description for the data, and click [Next] to continue
- Next, select the Coordinate System of the image file.
- Click the [Next] button to continue to the next step.
- A summary of the number of DXF entities processed and the number that will be created in Flex will be displayed
- Click the [Finish] button to finalize the import.

⚠️ Please Note: the coordinate system of the DXF file must be consistent with the one defined in the project settings as Flex will not perform a geotransformation on entities within the DXF. You can import points, polylines, or polygons that exist within a DXF file separately.

4.2 Importing VMOD/MODFLOW Models

Importing Visual MODFLOW Classic Projects
Before You Start!

If you need to modify or maintain a model that utilizes any of the following features, you must continue to use Visual MODFLOW Classic interface for this:

- Flow Engines (MODFLOW-96)
- Transport Engines (MT3D99, PHT3D)
- Flow Packages (ETS1, MNW, STR)
- MGO

To import your Visual MODFLOW Classic project, select the .VMF file and click OK to continue. The .VMF file must be from Visual MODFLOW v.2009.1 or later.

If your Visual MODFLOW Classic project contains Transport Boundary condition parameters, please refer to the discussion on the differences between how Classic and Flex handle point source concentrations for more details. As of Visual MODFLOW Flex version 5.0, transport boundary conditions will be imported along with flow boundary conditions.

While the model is created, you will see the corresponding inputs added to the Model Explorer.

Importing MODFLOW-2000, 2005 Data Sets

Often you may need to import an existing MODFLOW data set into Visual MODFLOW Flex in order to review the model, or evaluate different scenarios. Visual MODFLOW Flex can import existing MODFLOW data sets provided they are in MODFLOW-2000, MODFLOW-2005, or MODFLOW-SURFACT format.

Please Note: Visual MODFLOW Flex currently imports MODFLOW-2000 and MODFLOW-2005 Groundwater Flow process files only. As a result, it will ignore files associated with the Sensitivity Process, the Parameter Estimation Process, and the Transport Process. Support for these additional processes may be added to future versions of Visual MODFLOW Flex.

To import a MODFLOW-2000 or MODFLOW-2005 model data set, select the .NAM file (or .MODFLOW.IN) at the Import Grid screen.

Please Note: The .NAM file (or .MODFLOW.IN file) is an ASCII file containing a list of the input and output data files for the model, and their location (folder and pathname) on the
computer. If the model files have been moved to a new location, the file must be manually updated to reflect the new file location(s).

Once you have selected the desired model data file, click on the [Open] button to continue.

For more details on the Import MODFLOW process and limitations, please refer to Section 2.2 of the Visual MODFLOW (Classic) User Documentation.

⚠️ Please Note: Visual MODFLOW Flex does not currently support direct import of MODFLOW-SURFACT data sets. These must be imported as part of a Visual MODFLOW Classic project file.

### 4.3 Creating Surfaces

In Visual MODFLOW Flex, a surface refers to an attribute (e.g., elevation, conductivity, heads) represented as a set of continuous data over an area. Surfaces can either be imported directly (see "Importing Surfaces") from various file types using the import utility, or created by interpolating one or more points data objects. This chapter describes the process of creating surfaces from point data objects.

Surfaces are required in Visual MODFLOW Flex for defining the vertical boundaries of structural zones, e.g., horizons in a conceptual model. However, surfaces can also be used to assign spatially-variable attributes to property zones and boundary conditions, or for simply visualizing spatial variation using the 2D or 3D Viewers.

The create surface process allows you to generate surfaces using any numeric attribute in a point data object. The point data object can be one that has been imported using the import utility, or one that has been generated from other data objects, e.g., cross sections and wells. For more information on creating point data objects from well data and cross sections, please see Creating Points from Model Layer cross-sections and Converting well tops to points data, respectively.

This chapter presents information on the following topics:

- [Create Surface](#)
- [General Settings](#)
- [Interpolation Settings](#)

**Create Surface**

To create a new surface, follow the steps below:

- From the Data Explorer, right-click anywhere and select Create Surface... from the pop-up menu.
The Create Surface dialog will display. Specify the general settings, described in the following section.
General Settings

Enter a unique name for the surface in the Surface Name field, and a description of the surface (optional) in the Description text box.

The Data Source frame allows you to select the point data object(s) from which the surface will be created.

Adding a Data Source

- From the Data Explorer, select the desired Point data object from which the surface will be interpolated.
- Click the Add button, to add the point data object to the Data Source frame.
- Visual MODFLOW Flex allows you to create a single surface from multiple point data objects. Repeat the steps above to add additional points data objects to the Data Source frame.
- For each data source, select the Z Value from the combo box (shown below). The Z value can be any numeric attribute stored in the points data source, e.g., Elevation, Conductivity, Initial Concentration, etc.
Next, click the Interpolation Settings tab to define the interpolation settings for the surface. These settings are described in the following section.

**Interpolation Settings**

Select the interpolation method to use for generating the surface. Choose from the following interpolation methods:

- Inverse Distance
- Kriging
- Natural Neighbor
Below are brief descriptions of each interpolation method, taken from the GSLIB Geostatistical Software Library and User’s Guide (Deutsch and Journel, 1998). For a description of each interpolation parameter setting, click on the setting, and a brief description will be displayed beneath the interpolation settings grid.

Once the settings have been defined, click the [OK] button to generate the surface.

**Inverse Distance**

The Inverse Distance Squared method is very fast and efficient, weighted average interpolation method. The weighting factor applied to the data depends on the distance of the point from the grid cell, and is inversely proportional to the distance squared. Consequently, the greater the distance the data point is from the grid node, the smaller the influence it has on the calculated value.

The Inverse Distance Squared method for interpolation may generate patterns similar to the “bull’s-eye” surrounding points of observations. Selecting a larger number of nearest
neighboring data points may smooth this effect, but if the bull’s eye pattern is undesirable, then other methods for interpolation, like Natural Neighbor and Kriging, are recommended.

**Kriging**

Kriging is a geostatistical method that produces visually appealing maps from irregularly spaced data. Anistropy and underlying trends suggested in raw data can be incorporated in an efficient manner through Kriging. The program used, called kt3d, is available in the public domain from the Geostatistical Software Library (GSLIB), distributed by Stanford University, and is well documented by Deutsch and Journel (1998). The project kt3d performs simple Kriging, ordinary Kriging, or Kriging with a polynomial trend, and uses the standard parameter file used by GSLIP. If the semi-variogram components have already been modeled by the user, they can be incorporated into the program by choosing the appropriate set of parameters in the parameter file. The semi-variograms available include Spherical, Exponential, Gaussian, Power, and the Hoe effect models. If the variogram information is not available, the default linear variogram with no nugget effect should be used. This option is a special case of the Power model with the exponent equal to 1. When using the Kriging interpolation method, you may need to adjust the min and max radii of the variogram in order to get an ideal interpolation; these parameter values will depend on your data set. Default values are 100 and 100 along the major and semi-major axes. You will likely need to alter these values based on your input dataset and on the characteristics of the model domain.

**Natural Neighbors**

The Natural Neighbor method (Watson, 1994) is based on the Thiessen polygon method used for interpolating rainfall data. The grid node for interpolation is considered a new point, or target, to the existing data set. With the addition of this point, the Thiessen polygons based on the existing points are modified to include the new point. The polygons reduce in area to include the new points, and the area that is taken out from the existing polygons is called the “borrowed area”. The interpolation algorithm calculates the interpolated value as the weighted average of the neighboring observations where the weights are proportional to the borrowed areas. The Natural Neighbor method is valid only with the convex hull of the Thiessen polygon formed by the data points, and values outside the hull extrapolation should be used with caution.

The Natural Neighbor interpolation scheme may be visualized as a taut rubber sheet stretched to satisfy all the data points. The interpolated value at any location is a linear combination of all Natural Neighbors of that location, and the resulting surface is continuous with a slope that is also continuous. Combining the gradients or slopes with the linear interpolation provides results that are more smooth, and may anticipate the peak and valleys between data. Singularities and other undesirable effects may be lessened by incorporating the gradient factor.

The gradient influence on the results can be manipulated by two tautness parameters that you can enter. These parameters allow the interpolated surface to vary from purely linear interpolation to one which is well rounded and has a gradient factor. In all cases the slope discontinuities are removed and the resulting surface has slope continuity everywhere.
Defining an Interpolation Domain using a Polygon

When creating a surface, the interpolation domain is automatically calculated from the X and Y extents of the specified point(s) data set. There may be times when you do not want to use the entire points data set to generate a surface. In this case, you can manually define the interpolation domain (Xmax, Ymax, Xmin, Ymin) by specifying new values in the interpolation settings grid, or you can use an imported or digitized polygon data object.

- To define the interpolation domain using a polygon data object, follow the steps below:
  - In the Create Surface dialog box, select the Use a polygon extent check box.
  - Select a polygon data object from the Data Explorer, and then click the button.

- When the polygon data object is selected, the interpolation domain values in the settings grid will update with the X-Y extents of the selected polygon data object.
4.4 Creating New Data Objects

The 2D Viewer provides interactive drawing tools which allow you to create your own polygon, polyline and point data objects. This feature can be useful for digitizing boundary condition areas, property zones or your conceptual model boundary. To create a new polygon, polyline or points data object, follow the steps below.

- In the Data Explorer, right-click and select Create New Data Object from the pop-up menu. The following dialog will display:

![Create New Data Object Dialog]

- Select the Data Type from the combo box. You have the following options:
  - Point
  - Polyline
  - Polygon
  - DXF File (can only be visualized in 3D Viewer - can not be visualized in 2D Viewer tab or Layer View in Workflow tabs)
- Enter a name for the Data Name field.
- Click the [Ok] button to create the new data object.

Once created, the new data object will appear in the Data Explorer. From here, you can define the geometry of the data object using the 2D Viewer editing tools.

Digitizing & Editing Geometry in 2D Viewers

The 2D Viewer allows you to create and modify the geometry of points, polylines and polygon data objects. The process of drawing in a 2D Viewer is described below:

- Open a 2D Viewer by selecting Window from the Visual MODFLOW Flex main menu, and then clicking New 2D Window.
- Next, display the data object that will be edited in the 2D Viewer. Note: You can have multiple data objects displayed in the viewer while you’re editing/drawing the data object. However, make sure that the data object being edited is the “active” one by
selecting the data object name from the Layer combo box, located at the bottom of the 2D Viewer window.

- From the 2D Viewer sidebar, select the Pick button to set the 2D Viewer to pick mode. Pick mode allows you to click and select individual shape elements, e.g., vertices, line segments, features, that comprise the active data object.

- From the 2D Viewer sidebar, select the Edit button to set the 2D Viewer to edit mode. Once this button is selected, a set of editing buttons will display in the 2D Viewer sidebar. The edit buttons that show in the sidebar will vary depending on which type of data object is being edited. For example, the Add Points button will not be shown when you are editing/creating a polygon or polyline data object. The edit buttons are described below.

  **Add Points:** Digitize points in the 2D Viewer by moving the mouse cursor to the desired location and clicking the left mouse button. This button only shows when creating/editing a Points data object.

  **Add Polyline:** Digitize a polyline in the 2D Viewer. Click the left-mouse to start the line, and then left-click to insert a vertex along the line path. Double-click to end the polyline. This button only shows when creating/editing a Polyline data object.

  **Add Polygon:** Digitize a polygon in the 2D Viewer. Click the left-mouse button to start the polygon. Each successive left mouse-click will insert a vertex. Double-click to close the polygon.

  **Move Points:** Move a point by selecting and dragging the points to a new location in the 2D Viewer.

  **Rescale:** Select a shape element and stretch or shrink the geometry by selecting and dragging a side or corner of the blue box.

  **Rotate:** Select a shape element and rotate the geometry clockwise or anti-clockwise by selecting and holding the blue box, while moving the mouse.

  **Delete Shape:** Delete the selected shape.

  **Undo All:** Undo all edits. This button will revert the data object back to its original geometry.

If you are editing an existing polyline or polygon data object, the Selector combo box located at the bottom of the 2D Viewer allows you to select and modify the points (vertices) that comprise the features in the data object. For example, if you are editing a polyline data object, you can select Points from the Selector combo box, and each vertex that comprises each polyline will become active, allowing you to add, move or delete the vertices. When the Selector combo box is set to Points, the following icons are added to the 2D Viewer sidebar.

  **Add Vertex:** Add a vertex to a polyline or polygon feature by placing your mouse in the desired location on the line or polygon
boundary, and clicking the left mouse button.

**Move Vertex**: Select and hold the left mouse button and move the vertex to a new location the line or polygon boundary.

**Delete Vertex**: When selected, select a vertex to remove from a line and or polygon boundary.

Once you have created/modified the data object geometry using the edit tools described above, click the [End Edit button](#) to save the changes.

Finally, click the [View button](#) to return to the normal 2D Viewing mode.

### 4.5 Deleting Data Objects

To delete a data object, right-click on the data object from the Data Explorer, and select Delete from the pop-up menu.
5 Visualizing Data in 2D/3D

Visual MODFLOW Flex supports two types of interactive data viewers: 3D Viewer and 2D Viewer. The 3D Viewer is based on OpenGL graphics technology, allowing you to visualize graphically-rich three-dimensional representations of your data. The 2D Viewer allows you to view your data from a planar perspective, and provides various tools for editing and drawing data objects.

Visual MODFLOW Flex allows you to have multiple viewers opened and displayed simultaneously. Both viewers can be launched by clicking on Window from the main menu, and then selecting New 2D Window or New 3D Window.

This chapter presents information on the following topics:

- Creating Viewers, Opening, and Adding Data Objects
- Working With Viewers
- Viewer Controls
- Adjusting Viewer Settings
- 3D Viewer Animations
- Exporting Views

See Also:
- Style Settings for details on customizing how individual data layers are displayed
- Export for details on exporting data from a viewer
- Exporting 2D/3D Views to image

5.1 Creating Viewers, Opening, and Adding Data Objects

Opening a new 2D or 3D Viewer

There are two ways in which you can launch a 2D or 3D Viewer in Visual MODFLOW Flex: From the Main Menu or from the Data Explorer.

- From the main menu, select Window, then either 2D or 3D Viewer
- From the Data or Model Explorer, right-click on a data object and select 2D Viewer or 3D Viewer from the pop-up menu. A new viewer will then launch, displaying the selected data object. Please note that the viewers listed in the pop-up menu depend on which data object is selected (see table under “Displaying Data in Viewers”).

Adding Data Objects to the Viewer
To display data in a viewer, select the check box beside the data object in the Data Explorer or Conceptual Model Explorer. If multiple viewers are opened, the data will be shown in the active viewer (denoted by a blue title bar).

To remove data from a viewer, select the check box beside the data object so that it appears empty or “unchecked”.

⚠️ Please Note: some data objects may not be viewable in the 3D or 2D Viewers. (e.g. grid objects cannot be viewed in a 2D viewer)

When a data object is displayed in a viewer, it will appear as defined in the data object settings. For more information on viewing and modifying data object settings, please see Data Settings.
Layer ordering in 2D Viewer is determined by the sequence in which data objects are added to the viewer. For example, if data objects overlap each other, the data object added most recently will appear on top and effectively hide the underlying older data object(s).

You can bring layers to the top, by using the Layer combo box, located at the top of the 2D Viewer window.

The Layer combo box contains all of the layers currently displayed in
5.2 Working With Viewers

Using Multiple Viewers

Visual MODFLOW Flex allows you to have multiple 2D and 3D Viewer windows opened and displayed at one time. When a new viewer is opened, it is added to the window bar, located at the top of the main Visual MODFLOW Flex Window.

You can change the current active viewer by clicking on a different tab from the bar.

Docking/Undocking Viewers

Tabs can be undocked from the main window by drag-and-drop or by right-clicking on a tab and selecting "Undock Tab":

© 2019 by Waterloo Hydrogeologic
An undocked tab is displayed as a separate window that can be resized, minimized, or maximized using standard Windows controls. Multiple tabs can be undocked at the same time on multiple monitors, but at least one tab must remain docked in the main window. An undocked tab can be docked back into the main Visual MODFLOW Flex window by dragging it back on to the main window or right clicking on the window and selecting "Dock Tab".

**Flex Viewer**

The Flex viewer, (sometimes known as the Composite viewer), is available in the numerical modeling workflow for regular (MODFLOW Finite Difference) grids. This multi-purpose view gives you the ability to view and inspect your numerical model data along a selected layer, row, column, and/or 3D view.

- **Under Views**, select the various views you want to see in the Flex viewer; Visual MODFLOW Flex allows you to simultaneously show a layer, row, column and 3D Views. Place a check box beside the desired view and it will appear on the screen.
- Adjust a specific layer, row, or column using the up/down arrows. Alternatively, click on the button then click on any specific row, column, or layer in any of the 2D views, and the selected row, column, or layer will be set automatically.
- The 3D Viewer will show a "ThreeSlice" view of the selected Layer, Row, and Column; for details on controlling the display of this, see [3D Gridded Data Settings](#).
Objects in View

Below the list of Views you will see the Objects in View Panel.
This panel provides a list of the data objects that are visible in the current view; any objects that have been "checked on" from the Data tab or Model Explorer will be listed in this Objects in View; since both the Data tab and Model Explorer can grow in size and complexity as your project evolves, this Objects in View panel provides a nice short-cut to hide objects you don't wish to see in the view. In addition, you can access context menu options from this panel; these are menu items that were typically only available by right-clicking on the Data tab or Model Explorer.

Toolbar options (from left to right):

- **Settings**: loads the Settings options for that data object
- **Isolate**: temporarily selects that data object in the view and hides all others; you need to click and hold down this menu option in order to maintain the selection
- **Delete** (remove): removes this object from the view, but does not delete the underlying data from the project
- **More Actions**: Loads the similar context menu options that you would get if you were to right click on the object from the Model Explorer (or Data Tab); an example is shown below for "Conductivity"
5.3 Viewer Controls

The viewer controls allow you to interact with displayed data objects. The controls are accessible from the toolbar located along the right side of the viewer window.

- **View Mode**: The default mode. Allows you to zoom, rotate and pan the displayed data objects.
- **Pick Mode**: (2D Viewer Only) Allows you to select individual data object elements currently displayed in the viewer. When in Pick Mode, the Edit button will be shown in the sidebar, allowing you to edit the selected data object element.
- **Rotate**: Allows you to rotate the displayed data objects using your mouse. Click-and-hold on the displayed data, and move the mouse in a direction to rotate the data. (Available in 3D Viewer only).
- **Move/Pan**: Allows you to move/pan the displayed data objects in the viewer.
- **Zoom In**: Allows you to zoom in on the displayed data objects.
- **Zoom Out**: Allows you to zoom out of the displayed data objects.
- **Zoom Into box**: Use the mouse cursor to draw a box around an area of interest, and automatically zoom into this area.
- **Reorder layers**: Allows you to move layers up/down in a 2D view. Note that the topmost layer is the active layer; when doing editing of shapes, you must first set the desired layer to be the topmost.
- **Cell Inspector**: (2D Flex Viewer Only) Opens the Cell Inspector window.

**Linking 2D Viewer with Attributes Table**

The 2D Viewer can be linked with the spreadsheet table such that when a polyline, polygon or point feature is selected in the 2D viewer, its corresponding attribute data is automatically
highlighted in the spreadsheet table. Likewise, when an attribute row is selected in the spreadsheet table, its corresponding feature is highlighted in the active 2D Viewer. In order to have this bidirectional linking between viewer and spreadsheet table, the 2D Viewer must be in Pick mode, and the spreadsheet table must be opened.

**Tip!** Data object spreadsheet tables can be viewed by right-clicking on the data object in the Data Explorer, and selecting Spreadsheet... from the pop-up menu. 2D Viewers can be set to pick mode by selecting the Pick Mode button from the viewer sidebar.

### 5.3.1 Cell Inspector

The Cell Inspector is available when a 2D view (i.e. Layer, Row, Column) is active in a Flex Viewer associated with either a regular finite difference grid or unstructured grid Numerical Model Workflow step.

Clicking the icon will open a Cell Inspector for the numerical model associated with the active workflow step. The title bar of the Cell Inspector will include the name of the numerical to which it applies. You can open one cell inspector for each numerical model in your project. The Cell Inspector window consists of two tabs Cell Values and Select Items.
Cell Values Tab

The Cell Values tab displays the values (in group view or list view, see table above) for selected items applicable to the most-recently queried cell in the relevant model.

Values in the Cell Values tab are read-only and cannot be modified. The one exception to this is cell address value(s). For finite difference grids the cell address consists of the Layer, Row, and Column indices (i,j,k), while in unstructured grids, the cell address consists of a single global cell index (i). Editing these values will refresh the Cell Values tab with the values associated with the newly entered cell address.

Controls

The following controls are part of the Cell Values tab:

- **Refresh**: re-queries the values based on the current cell address.
- **View Type**: Allows you to select either group view or the list view. The group view is the default view and presents cell data in hierarchical groups (e.g. Input > Conductivity > Kx). The list view presents all information in tabular format with each hierarchical level given its own field and facilitates copying the data to external formats (such as spreadsheets).
- **Expand All**: Allows you to expand all grouped nodes in the Cell Values Tab
- **Collapse All**: Allows you to collapse all grouped nodes in the Cell Values Tab
- **Select All**: Allows you to select all values in the Cell Inspector Cell Values Tab
- **Copy Selection**: Allows you to copy the selected cell(s) to the Clipboard for pasting the selected values into an external application (i.e. spreadsheet or text editor)
- **Time Indicator**: Indicates the time step/stress period applicable to the cell inspector and is based on the value shown in the time picker for the active view. Times are shown in relative time

Select Items Tab

The Select Items tab allows you to select which model items (e.g. grid properties, model properties, or boundary conditions) will be displayed in the values tab.
Controls
The following controls are part of the Cell Values tab:

- **Refresh**: refreshes all data layers in the model explorer and adds/removes groups based on newly created/deleted properties, boundary conditions, and/or model results
- **Expand All**: Allows you to expand all grouped nodes in the Select items Tab
- **Collapse All**: Allows you to collapse all grouped nodes in the Select items Tab
- **Select All**: Allows you to select all model data nodes
- **Select None**: Allows you to clear all selections
- **Show Selected**: Allows you to show only the selected items and hide the deselected items
- **Show Deselected**: Allows you to show only the deselected items and hide selected items
- **Search**: Allows you to type text that will be used to search and filter items
- **Clear Search**: Allows to clear the search and/or show all hidden items

5.4 Adjusting Viewer Settings
The following settings are available in both 3D and 2D Viewers:

- **Changing the Background Color**: To change the background color of a 2D or 3D Viewer, right-click anywhere within the viewer, and select Background Color from the pop-up menu. The Color combo box will display on your screen. Select a new color and then click the [OK] button.
- **Showing the Viewer Axis**: To show or hide the viewer axis, right-click anywhere within the viewer, and select Axis from the pop-up menu.

The following settings only apply to 3D Viewers.

- **Setting the Vertical Exaggeration**: The Vertical Exaggeration is the ratio of the scale of the Y-axis to the scale of the X-axis. Vertical exaggeration can be used for discerning subtle topographic features or when the data covers a large horizontal distance (miles) relative to the relief (feet). By default, the vertical exaggeration is set to 0. You can change the vertical exaggeration using the Exaggeration text box, located at the bottom of the 3D Viewer (shown below).
• **Resetting the Viewer**: To zoom out to the full extents of your data click the Reset Scene Position button located at the bottom of the viewer.

  **Please Note**: clicking this button will reset the rotation and zoom level back to the original view configuration.

**3D Viewer - Creating Cutaways**

Visual MODFLOW Flex allows you to remove portions of the model from the 3D Viewer by creating cutaways. To create a cutaway in a 3D-Viewer, follow the steps below:

![CutAway Properties dialog box](image)

- Right-click anywhere inside the 3D Viewer, and select CutAway Properties from the pop-up menu. The following dialog box will display on your screen.
- Click the CutOffs Active check box to enable CutAways.
- Under the Activity tab, specify which slices to make active by clicking the appropriate Active check boxes. The YZ slice refers to a vertical plane along the Y and Z axis, the XZ slice refer to a vertical plane along the X and Z axis, and the YZ slice refers to a horizontal plane along the Y and Z axis.
- For each active slice, select which portion to remove from the viewer (for example, the portion of the model that is left or right side of the slice, before or after the slice, or above or under the slice).
• By default, the position of each slice is automatically positioned in the middle of the data along its respective axis. However, you can manually change the position of the slice by entering a new value in the Position field.

Please Note: the Xmin, Xmax, Ymin, Ymax, Zmin and Zmax fields are read-only, and cannot be changed.

• Alternatively, you can set the position of a slice by entering a value in the Fraction field. For example a fraction value of 0.5 will set the slice position in the middle of the 3D data.
• Click the [OK] button to apply the changes to the current 3D Viewer.

3D Viewer Performance Preferences

For details on adjusting the 3D View performance settings, see 3D Viewer Preferences.

5.5 3D Viewer Animations

The 3D Viewer can animate items and also record static or transient models to .AVI file.

Right-click on the 3D viewer and select the Animation Parameters. The following options are available:
Rotate will allow you to rotate the model domain around the specified axis. If you wish to save to a video file, select the check box beside Write Video File and specify a name and directory. The combo box below this allows you to specify the video resolution.

If you click on the Time tab, you will have options to specify the start and end time and the number of time frames. These options are only appropriate if you are animating Heads or Drawdown that has multiple output times.
Click OK after defining the desired Animation settings.

Then right-click on the 3D viewer and select Animate and the rotation will start. If you have selected to record a video file, then the video recording will begin.

Once you are done recording, right click on the 3D viewer again and select Animate to stop the animation.

The AVI recording can be memory and CPU intensive. The performance will depend on the size of the model you are visualizing and the hardware on your computer (memory, CPU, video driver).

Please be patient while this is recording. The recording time will also increase when you include 3D gridded objects, such as Conductivity Zones, calculated Heads, Drawdown, or Concentrations.
5.6 Exporting Views

To export a 2D Viewer to a graphics file, e.g., *.BMP, *.TIF, *.GIF, *.JPEG, follow the steps below:

- Right-click anywhere within the viewer
- Select Export Current View to Image from the pop-up menu.
- A Save As window will display, allowing you to specify the location on your computer where the graphics file will be saved.

To export a 3D Viewer to a graphics file, follow the steps below:

- Right-click anywhere within the viewer
- Select Save as image from the pop-up menu.
- The following dialog box will display on your screen:

![Save as Image dialog box]

- Select the desired image size from the Image Size combo box. If you select Custom, then specify the desired image dimensions in the Height and Width combo boxes.
- Click the [...] button and specify a folder location on your computer to save the image file.
- Click the [Ok] button to save the image.
6 Data Settings

Visual MODFLOW Flex allows you to view and modify various settings and properties for each imported or digitized data object. In general, data object settings consist of the following categories:

- **General**: View data object metadata including source data information, statistics, native coordinate system, field mappings; view the attribute and geometry table.
- **Operations**: Apply arithmetic and geometric operations to data object geometry and attributes.
- **Style Settings**: Modify various style settings for different data object elements, e.g., point/line symbology, show labels, color rendering by attribute, create isosurfaces, set transparency etc.

Data object settings can be accessed from the main Visual MODFLOW Flex window. To view the settings for a data object, in the Data Explorer, right-click on the desired data object and select Settings... from the pop-up menu.

This chapter presents information on the following topics:

- Viewing **General Settings**
- Viewing the **Data Table**
- Performing **Data Operations**
- Modifying **Style Settings**
6.1 General Settings

General settings consist of data object meta data including the coordinate system, field mappings, and source data information. General settings can be accessed by expanding the General node in the Settings tree. The settings in each sub-node are described below.

**Title Bar**

The title bar of the General Settings window will include the name of the data object and the viewer in which the style settings apply.

**Info**

The Info node contains the following information:

- **Name**: The name of the data object (read only) defined during import.
- **Type**: The type of the data object, e.g., points, polygon, polyline.
- **Data Source**: The folder path of the data source when the data was imported.
• **Color**: The color of the data object when displayed in 2D and 3D Viewers. Click the color box to select a new color for the data object.

• **Description**: The description of the data object as defined during import. This field is editable allowing you to update the description at any time.

Table (Button): The table button allows you to view the geometry and attributes of the data object in a table view. For more information on the table view, please see "Viewing the Data Table" section.

**Details**

The Details node contains information on the coordinate system of the data object, and the field mappings specified during import.

**Statistics**

The Statistics node displays a count summary of the various data elements that comprise the data object. For example, for polygon data objects, the statistics node will show the number of polygons, polygon parts and vertices in the data object. For cross section data objects, the number of wells and cross sections is displayed.

For Property and Structural Zone conceptual model objects, various statistics are automatically calculated and displayed under the statistics nodes, including: Min and Max X, Y and Z values, Area and Volume.

### 6.2 Data Table

**General**

The data table allows you to view the geometry and attribute values of a data object. The data table can be accessed from the General settings in the Settings dialog, or it can be launched by right-clicking on the data object in the Data Explorer, and selecting Spreadsheet... .

ℹ️ **Please Note:** the Map and Cross Section data objects do not have a data table.
The Attribute tab contains the attribute data of the selected data object. Each column in the attribute tab represents an attribute. The Geometry tab contains the geometry (X,Y, Z) values of the data object. In both tabs, the FID column uniquely identifies each feature in the data object.

To copy data to the Windows clipboard, highlight the data to be copied, and then click the Copy button, or press CTRL+C on your keyboard.

Tip! When a data object is shown in 2D Viewer and the viewer is set to Pick Mode, you can select a row from the attribute or geometry table, and the corresponding feature will be highlighted in the 2D Viewer.

Modifying Attribute and Geometry Data

Imported geometry and attribute data can be modified in the Data Table. To make changes to data, click the Begin Edit button to enter edit mode. Make the necessary changes to the data table and once finished, click the End Edit button to save the changes. Please note that the data table only allows you to modify existing attribute and geometry data. Currently, Visual MODFLOW Flex does not allow you to create new columns, i.e., new data object attributes. This can only be done during the data import process.

Well Table

For Wells data objects, the data table is different than that of other data objects. The Well data table is designed to allow you to add and/or modify wells and associated well data, e.g., pumping schedule, screen intervals, observation points, etc.

To access the well table, right-click on a Wells data object in the Data Explorer, and select Settings... . In the Settings dialog, click on the Table button.
In the Well Table, there are two tabs: Vertical and Horizontal. Each tab is described in the following sections.

**Vertical Wells**

The Vertical tab allows you to view and modify data for vertical wells.

The Well Heads table contains a list of all the wells in the data object. The data stored in this table includes the Name, X-Y coordinates, Elevation and Depth for each well. When a well is selected, its corresponding attribute data is displayed in the adjacent data tables, e.g., Screens, Pumping Schedule. You can search for a well in the Well Heads table by entering the well name in the text box, located at the top of the window, and then clicking the [Find] button.

The Data to Display list box allows you to select which tables to display. For example, if Screens is selected (default), the Screens and Pumping Schedule tables will be shown. If Observations is selected, the Observation Points and Observation Data tables are shown. If Well Tops is selected, the Well Tops table will be shown.

The Display Format frame allows you express the Z values in the data tables as either an Elevation or a Measured Depth (with respect to the well head Zmax).
At the top of each table, there is a set of buttons that allow you to add, remove and modify the contents. The function of these buttons, in order from left to right, is described below:

- Add a row to the table
- Insert a row above the active row
- Insert a row below the active row
- Remove the active row from the table

**Adding Well Head Data**

To add an item to the Well Head table, follow the steps below:

- Click the Add Row button from the Well Head toolbar to add a new item to the table.
- Enter an alphanumeric name in the Well Name column. Note: The well name must be unique and it may contain hyphens and spaces, but not the forward or backward slash characters.
- Enter the X-Y coordinates of the well head in the X and Y fields, respectively.
- Enter the elevation of the well head in the Zmax field.
- Enter the depth of the well in the Zmin field.

**Adding Well Screens**

To add an item to the Well Screen table, follow the steps below:

- Make sure the Screens option is selected from the Data to Display box.
- Select a Well from the Well Head table.
- Click the Add Row button from the Screens toolbar to add a new item to the table.
- Enter a screen identification number in the Screen ID field.
- Enter a screen top elevation (or measured depth) in the Screen Top field.
- Enter a screen bottom elevation (or measured depth) in the Screen Bottom field.

**Adding/Editing Pumping Well Schedule**

The Pumping Schedule table is used to enter the well pumping rates for specified time periods. Negative pumping rate values are used for extraction wells, and positive pumping rates are used for injection wells.

Pumping well schedules are defined for the entire well, and therefore a screen must exist before a pumping schedule can be defined.

To add pumping schedule items to the pumping schedule table:

- Make sure the Screens option is selected from the Data to Display box.
- Select a Well from the Well Head table, and a Screen from the Screen Table (if multiple screens exist).
- Click the Add Row button from the Pumping Schedule toolbar to add a new item to the table.
- Enter a Start time value and press the <Tab> key to advance to the End time field.
Enter an End time value and press the <Tab> key to advance to the Rate field.
- Enter a pumping Rate value (remember to use a negative value for extraction wells)
- Press the <Tab> key again to create a new schedule item.
- The final time in the pumping schedule should have a pumping rate of 0 to indicate the stop time.

**Please Note:** If the pumping schedule is not specified for the entire length of the transient simulation, then it will assume the well is shut off for the time where no information is available. For steady-state simulation, the pumping rate for the first time period will be used as the steady-state pumping rate.

The “Show as Relative Times”, when selected will display time schedule data starting at 0, and relative to the start date defined in the Modeling Objectives; this is useful when you want to correlates the raw well data to MODFLOW time formats. When this option is not selected, then time schedule data will appear in “Absolute” (calendar) time formats.

**Adding Observation Points**
Observation Points are the elevations at which head or concentration observations are recorded. Although most monitoring or observation wells are installed with a well screen spanning a known interval of the aquifer, Visual MODFLOW Flex requires a single observation point elevation to be defined instead of a well screen interval.

To add an observation point for a well:

- Select the Observations option from the Data to Display box. This will show the Observation Points table.
- Select a well from the Well Heads table
- Click the Add Row button from the Observation Points toolbar to add a new item to the table.
- Enter an observation point ID in the ID field.
- Enter an elevation value in the Elevation field.

The MODFLOW simulator supports head and concentration observation wells with multiple observation points throughout the length of the well-bore. Repeat the steps above to add additional points.

**Adding Head Observation Data**
The Observation Data table is used to enter the observed values at specified times, for the selected observation point.

To add observations to the Observation Data table, follow the steps below:

- Select Observations from the Data to Display box.
- Select the well from the Well Heads table for which observation data will be added.
- Select the desired observation point from the Observation Points table.
• Click the Add Row button from the Observation Data toolbar to add a new item to the table.
• Enter the time at which the head was observed in the Time field.
• Enter the observed head values in the Head field.

The "Show as Relative Times", when selected will display time related data starting at 0, and relative to the start date defined in the Modeling Objectives; this is useful when you want to correlates the raw well data to MODFLOW output time. When this option is not selected, then time related data will appear in “Absolute” (calendar) time formats.

**Adding Well Tops**

The Well tops table is used to enter the elevation points along the well path, where the well intersects with a horizon. Well top information can be used in Visual MODFLOW Flex to create surfaces which can then be used to define conceptual model horizons. For more information on creating surfaces from well tops, please see "Converting Well Tops to Points Data Object" section.

To add well top information to the Well Top table, follow the steps below:

• Select Well Tops from the Data to Display box.
• Select the well from the Well Heads table for which well top data will be added.
• Click the Add Row button from the Well Tops toolbar to add a new item to the table.
• Enter the location of the well top as a measured depth in the Depth field.
• Enter the name of the formation, e.g., Clay, Sand etc., in the Formation field.

**Deviated Pumping Wells**

Deviated pumping wells can be defined by specifying flux boundaries in the corresponding cell sets to represent the wells.

### 6.3 Data Operations

For most data object types, Visual MODFLOW Flex allows you apply various arithmetic operations to your source data. Operation settings can be accessed by clicking on the Operations tree node in the Settings window. (To access the Settings window, in the Data Explorer, right-click on the data object and select Settings... from the pop-up menu).
Visual MODFLOW Flex supports the following data operations:

- Arithmetic Operations: Polygons, Polylines, Points, Surfaces, and Maps only
- Attribute Operations: Surface, Points, Time Schedules only
- Converting Model Layers to Points Data: Object Cross Sections only
- Converting Well Tops to Points Data: Wells Objects only
- Calculating Well Head Elevation (Z) from a Surface: Well Objects only

Each type of operation is described in detail in the following sections.

**Arithmetic Operations**

Arithmetic operations allow you to shift the elevation values in the data object source data according to a user specified arithmetic expression. For example, you can use this option to drape a map over a specified surface data object. You can also shift polylines, polygons,
points or surfaces up or down by a specified constant value. Arithmetic operations can be applied to polygons, polylines, points, surfaces and maps.

When the Arithmetic node is selected from the Settings tree, the following dialog will display:

To apply an arithmetic operation follow the steps below:

- Select the desired arithmetic expression from the Select Operation combo box.
- If you are unsure of what the expression does, refer to the provided description in the Description and Instructions text box.
- The contents of the Input Parameters frame will vary depending on the selected expression.
- If the selected expression contains a constant value, e.g., \( Z = \text{Constant} \), enter a value in Value field.
- If the selected expression requires a surface, e.g., \( Z = \text{Surface}(x,y) \), then select the desired surface from the Data Explorer, and then click the button to insert the surface into the Value field.
- Optional: Select the Save As New Data Object check box to save the transformed data as a new data object.
- Click the [Execute] button to apply the operation.

Please Note: If the data object is being viewed in a 3D Viewer while the operation is applied, you may have to turn off the data object, and then turn it back on to see the changes.

Attribute Operations

Modify an Attribute using a Constant Value

For time-schedule data objects, Visual MODFLOW Flex allows you to modify attribute values using a specified constant value. For example, the constant value can be set equal to, added to, subtracted from, and multiplied by the existing attribute values.
To apply an attribute operation:

- Select the desired expression from the Select Operation combo box.
- If you are unsure of what the expression does, refer to the provided description in the Description and Instructions text box.
- In the Input Parameters frame, select an attribute from the combo box under the Value column.
- Enter a value in the Constant field, under the Value column.
- Optional: Select the Save As New Data Object check box to save the transformed data as a new data object.
- Click the [Execute] button to apply the operation

**Creating an Attribute from 3D Gridded Data Object - For Points and Surface data objects only**

This operation allows you to create a new attribute using 3D Gridded data for surface and points data objects. This feature can be useful after you have run the numerical model simulation using Visual MODFLOW Flex, and you have imported the .HDS file back into Visual MODFLOW Flex as a 3D Gridded data object for visualizing the heads in 3D Viewer. The head information in the 3D Gridded data object can be extracted, and interpolated for a surface or points data object. You can then use the Color by Attribute feature to display the heads information on the surface or points data object. Likewise, this can be used for
visualizing any attribute contained in a 3D Gridded data object on a surface or points data object. This procedure is described below.

This operation can be accessed from the Settings dialog. Select the surface or points data object in the Data Explorer, right-click and then select Settings... Once the Settings dialog launches, expand the Operations node, and select Attribute from the settings tree.

- Select Create new Attribute from the Select Operation combo box.
- Select the 3D Gridded data object from the Data Explorer, and select the button to insert the data object into the dataObject field.
- Once the 3D Gridded data object is selected, its available attributes are populated in the Attribute combo box. Select the desired attribute from the Combo box.
- Optional: Select the Save As New Data Object check box to save the transformed data as a new data object.
- Click the [Execute] button to apply the operation.

Once the operation is applied, you can confirm that the new attribute was created by viewing the table view for the selected surface or points data object.

**Converting Model Layers to Points Data Object**
For Cross Section data objects only
This operation allows you to create a new points data object, for each model layer interpretation, from all cross sections in the data object that include this interpretation. Once the points data objects are created, you can then create surface data objects, which can then be used to define the horizons of your conceptual model.

**Please Note:** This feature is available for model layer interpretations only.

To create points data objects from cross section interpretation model layers:

- From the Select Operation combo box, select Convert Model Interpretations to Points Data Object (default).
- Click the [Execute] button to apply the operation.
- Once the points data objects are created, they will be added to the Data Explorer, where they can be used to create surface layers.

**Converting Well Tops to Points Data Object**

*For Wells data objects only*

This operation allows you to create a new points data object, for each well top formation in a wells data object. The resulting points data objects can then be used to create surfaces, which can be used to define the horizons of a conceptual model.
Well top data can either be included during data import, or they can be manually defined in the well table view. For information on defining well tops, please see "Adding Well Tops" section.

To access this operation, right-click on the desired wells data object in the Data Explorer, and select Settings... from the pop-up menu. In the Settings dialog, select the Operations node, and a window, similar to the one shown below, will display.

To create new points data objects from well top formation, follow the steps below:

- Select the Convert well tops to points data objects option from the Select Option combo box (selected by default).
- Click the [Execute] button to apply the operation.
Once the points data objects are created, they are added to the Data Explorer using the naming convention [wells data object name]_[formation label] (shown above).

**Calculating Well Head Elevation (Z) from a Surface**

For well data objects only

This operation is only available for well data objects. It allows you to calculate elevation values for each well head in the data object, using a specified surface data object. Please note, any well head elevations that have been added manually or imported will be overwritten with the elevation values calculated from the specified surface.
To calculate well head elevation from a surface,

- Select Calculate well head elevation (Z) from a surface from the Select Operation combo box.
- Select the desired Surface data object from the Data Explorer, and select the button to insert the data object into the surface field.
- Click the [Execute] button to apply the operation.

Once the operation is applied, you can confirm that the new Z values were created by viewing the table view for the selected well data object.

Please note that Visual MODFLOW Flex will ignore wells where the elevation of the bottom of the well (Zmin) is greater than the calculated well head elevation.

### 6.4 Style Settings

Visual MODFLOW Flex provides you with a wide variety of style settings, allowing you to modify the appearance of data objects in both 3D and 2D Viewer. The style settings can be accessed by expanding the Style node in the Settings tree (shown below).
Please Note: the Style node will only be available when the particular data object is being shown in 2D or 3D Viewer.

In general, each data object has its own set of style settings, although some settings are common between data objects. The following sections describe the style settings for the various data objects.

Select the links below to jump to the appropriate sections:

- Points, Polylines, and Polygons
- Cross-Sections
- Wells
- Surfaces
- 3D Gridded Data

6.4.1 Points, Polylines, and Polygons

Points/Vertices

The style settings for points data objects are described below:
Select the color method, symbol, and size from the appropriate combo boxes.

The Show in Cutaway check box allows you to show points or vertices in cutaway regions in the 3D Viewer window. When this option is disabled, points or vertices will not show in any areas that have been “hidden” in the 3D Viewer window by creating “Cutaways”. For information on creating cutaways, please see "Adjusting Viewer Settings" section.

Choose from the following color methods:

- **Specified**: Points are colored using the color specified in the General settings.
- **By Attribute**: Points are colored based on a specified data object attribute as explained below.
**Color By Attribute**

Data objects can be colored based on a specified attribute. Color rendering can be applied to any shape element that contains attributes. To color a data object by attribute, follow the steps below.

In the Settings for the selected data object,

- Expand the Style node and select the shape element to be colored, e.g., Points
- From the Color combo box, select the By Attribute option.
- From the Settings tree, select the Colors node and the following dialog will appear.

- Select the desired attribute from the Attribute combo box. You will notice that the min and max values are displayed to the right of the combo box. Some data objects will have attributes while others will not. The following table lists which data object types
can have attributes available for color rendering:

<table>
<thead>
<tr>
<th>Data Object</th>
<th>Available Attributes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Points, Polygons,</td>
<td>Only if attributes were created during import</td>
</tr>
<tr>
<td>Polylines</td>
<td></td>
</tr>
<tr>
<td>Surfaces</td>
<td>Z</td>
</tr>
<tr>
<td>3D Gridded Data</td>
<td>Z and other attributes that were added during import</td>
</tr>
<tr>
<td>Conductivity</td>
<td>ZoneID, Kx, Ky, Kz</td>
</tr>
<tr>
<td>Storativity</td>
<td>ZoneID, Ss, Sy, Tp (total porosity), Ep (effective porosity)</td>
</tr>
<tr>
<td>Initial Heads</td>
<td>ZoneID, Initial Heads</td>
</tr>
<tr>
<td>Recharge</td>
<td>ZoneID, Recharge Rate, Ponding Depth</td>
</tr>
<tr>
<td>Evapotranspiration</td>
<td>ZoneID, Evapotranspiration Rate, Extinction Depth.</td>
</tr>
<tr>
<td>Heads</td>
<td>Head</td>
</tr>
<tr>
<td>Drawdown</td>
<td>Drawdown</td>
</tr>
</tbody>
</table>

- Select the Color Classification: by default, the colors will be calculated from the min and max values for the current data object. Visual MODFLOW provides an option to use Project-wide Color Palettes. This is useful when you have multiple data objects that are rendering the same attribute (e.g. heads from different model runs, conductivity distributions), and you want to make qualitative comparisons between these. This is challenging when each data object has its own min and max values and are colored based on this. However this becomes much easier when these data objects all read from a common color palette. If this option is checked on, you can access the color palette and adjust min and max values. See Project Color Palettes for more details.

- For the selected attribute, you can define various settings. These settings are described below.

  **Type**: Select between Stretched or Classified. The stretched option allows for color shading, i.e., continuous color gradient. The classified option allows for discrete intervals, i.e., zebra, in the color gradient.

  **Classes**: There are two options for defining the number of classes to be used: Number of Classes and Equal Intervals.

  **Number of Classes**: Specify the number of color classes in the text box.

  **Equal Intervals**: Specify an interval spacing, and the number of classes will be determined from the range of the attribute data. For example if your data range is 100, and you specify a 10 interval spacing, 10 classes will be created.
As the number of intervals/classes is modified, rows are automatically removed or added to the color table. For the Elevation color scheme, the maximum value will always be Red and the minimum value will always be Blue, but the gradient between will change automatically depending on the specified number of classes.

- Visual MODFLOW Flex automatically calculates the intervals based on the defined specified number of classes or equal intervals.
- Once the settings have been defined, click the [Apply] button to show the changes in an active 2D or 3D Viewer.

**Area**

<table>
<thead>
<tr>
<th>General</th>
<th>Operations</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Style</td>
</tr>
<tr>
<td>Are</td>
<td>General</td>
</tr>
<tr>
<td></td>
<td>Labels</td>
</tr>
<tr>
<td></td>
<td>Colors</td>
</tr>
<tr>
<td>Lines</td>
<td>Vertices</td>
</tr>
</tbody>
</table>

Select the Show Area Fill check box to show/hide the area file. If unchecked, only the shape boundary will be visible in 2D and 3D Viewers.

Select the color method and fill pattern symbology, from the appropriate combo boxes.
Choose from the following color methods:

- **Specified**: Shape is colored using the color specified in the General node.
- **By Attribute**: Shape is colored based on an attribute. See "Color By Attribute" section for more information on color rendering.
- **Custom**: Specify a color for the area fill. This color will overwrite the default color defined in the general settings for this particular shape element.

Select the Transparent checkbox to make the polygon fill pattern transparent. Use the adjacent Transparency text box to set the level of transparency, e.g., a higher value will make the fill more transparent.

The Show in Cutaway check box allows you to show areas the polygon in cutaway regions in the 3D Viewer window. When this option is disabled, the polygon will not show in any areas that have been “hidden” in the 3D Viewer window by creating “Cutaways”. For information on creating cutaways, please see "Creating Cutaways”.

**Lines**

Select the Color method, line Pattern (solid or dash), line Width from the appropriate combo boxes.
Choose from the following color methods:

- **Specified**: Line is colored using the color specified in the General settings.
- **Custom**: Specify a color for the line element. This color will overwrite the default color defined in the general settings for this particular shape element.

The Show in Cutaway check box allows you to show areas the line in cutaway regions in the 3D Viewer window. When this option is disabled, lines will not show in any areas that have been “hidden” in the 3D Viewer window by creating “Cutaways”. For information, please see the "Creating Cutaways" section.

**Labels**

The following label settings are available under the General node (shown above):
- **Show Labels**: Show/Hide the labels in 2D/3D viewer.
- **Label Field**: This combo box contains all attributes for the selected data object. Select the desired attribute field to use for the labels.
- **Font**: Select the label font from the combo box.
- **Size**: Set the text size of the labels.
- **Style**: Set the font style for the labels, e.g., Regular, Bold, Font, Italicized, etc.
- **Color**: Set the color of the label text.

**Format**

The following label settings are available under the Format node:

- **Format**: Choose between Numeric or Scientific notation.
- **Decimals**: Set the number of decimals to plot for each label.

**Placement (Polylines only)**

- **Position**: Display the label above, below, or on the line.
- **Offset**: This parameter controls how far the label will be placed from the line.
- **Location along the line**: Display the label at the start, in the middle or at the end of the line.
- **Orientation to the line**: Display the label parallel, perpendicular or horizontal to the line.

### 6.4.2 Cross Sections

The following section describes the available style settings for cross-section data objects.

To access the style settings, right-click on the cross-section data object in the Data Explorer, and select Settings... from the pop-up menu. Then, in the Settings dialog, expand the Style node to view the style settings.
cross-section data objects consist of two main elements; the interpretation layers and the cross-section wells. The settings for each element can be accessed by clicking on the Interpretation or Wells node, respectively.

**Interpretation**

A screen capture of the interpretation settings is shown above. From the Interpretation Type combo box, select which interpretation layer to show in 3D Viewer. Select from Model, Geology or Hydrogeology.

When an interpretation layer is selected from the combo box, its associated cross-sections are listed in the grid below. Under the Visible column, select which cross-section to show/hide in 3D Viewer.

Select the Show All check box to show all the cross-sections for the selected interpretation layer.

Select the Show Labels check box to show the label for each cross-section.

**Wells**

The wells node contains settings for changing the appearance of the cross-section wells. These settings are described below.
- **Show Wells**: Check this option to show the well geometry.
- **Show Labels**: Check this option to show the well label above each well.
- **Line Style**: Select the type of line to display. Choose between solid or dashed.
- **Line Width**: Specify the width of the wells.
- **Color**: Change the color of the wells.

Click the [Apply] button to display the changes in an active 3D Viewer window.
6.4.3 Wells

The following section describes the available style settings for Wells data objects.

To access the style settings, right-click on the well data object in the Data Explorer, and select Settings... from the pop-up menu. Then, in the Settings dialog, expand the Style node to view the style settings.

The Wells settings are divided into three sections: Well Head, Well Path and Well Tops.

**Well Head**

The Well Head node provides options for changing the appearance of the well heads (top of the wells).

When the General subnode is selected, the following options are available:

- **Symbol**: Select the well head symbology from the combo box. Choose from various symbols including circle, square, cross, diamond etc.
• **Size**: Specify the size of the well heads.
• **Color**: Select a color the well heads.

Click the [Apply] button to view the change in an active 2D or 3D Viewer.

For information on the Labels node, please see "Labels" section.

**Well Path**

The Well Head node provides options for changing the appearance of the well path. These options are described below.

When the General subnode is selected, the following options are available:

• **Show Lines**: Select this option to show/hide the well path.
• **Type**: Specify the line type for the well path. Choose between a solid line or a dashed line.
• **Width**: Specify the width of the well paths.
• **Color**: Select a color the well paths.
Click the [Apply] button to view the change in an active 2D or 3D Window.

For information on the Labels node, please see the "Labels" section.

**Well Tops**

The Well Tops node provides options for changing the appearance of the well tops. These options are described below.

When the General subnode is selected, the following options are available:

- **Symbol**: Select the well top symbology from the combo box. Choose from various symbols including circle, square, cross, diamond etc.
- **Size**: Specify the size of the well top symbols.
- **Color**: Select a color for the well tops.

Click the [Apply] button to view the change in an active 2D- or 3D-Window.
For information on the Labels node, please see the "Labels" section

6.4.4 Surfaces

The following section describes the available style settings for Surface data objects.

To access the style settings, right-click on the surface data object in the Data Explorer, and select Settings... from the pop-up menu. Then, in the Settings dialog, expand the Style node to view the style settings.

Colors

The Colors node provides options for coloring the surface layer by elevation value. The following options are available:

- **Show Color Fill**: This option allows you to show or hide the color fill. If disabled, the surface will appear in the color defined in the General settings.
• **Transparency:** This option allows you to make the surface appear transparent. When the Transparent checkbox is selected, use the adjacent slider bar to set the level of transparency.

• **Type:** Select the type of gradient to use for coloring the surface. Select Stretched to use a continuous color gradient, or select Classified to use discrete color zones.

• **Classes:** There are two options for defining the number of classes to be used: Number of Classes and Equal Intervals.
  o **Number of Classes:** Specify the number of color classes in the text box.
  o **Equal Intervals:** Specify an interval spacing, and the number of classes will be determined from the range of the attribute data. For example if your data rage is 100, and you specify a 10 interval spacing, 10 classes will be created.

As the number of intervals/classes is modified, rows are removed or added to the color table. The maximum value will always be Red and the minimum value will always be Blue, but the gradient between will change automatically depending on the specified number of classes.

Click the [Apply] button to view the changes in an active 2D- or 3D-Window.

**Contour Lines**
The Contour Lines node provides options for showing contour lines on the surface layer. The following options are available:

- **Show Contour Lines**: Show/Hide the contour lines.
- **Show Contour Labels**: Show/Hide the contour labels.
- **Contour Interval**: Set the contour increment value.
- **Start Contour Value**: Value above which automatic contours will be displayed
- **Finish Contour Value**: Value below which automatic contours will be displayed
- **Use Custom Contour Levels**: Show/Hide the custom contour values specified in the Custom Contour List window
  - **New Value**: Value to be added to the Custom Contour List window
Add New: Add the "New Value" to the Custom Contour List window. Specified values must be within the range of the active dataset.

Remove: Removes the highlighted value from the Custom Contour List window

- **Line Style**: Select the contour line style from the combo box. Choose between solid line or dashed line.
- **Line Width**: Set the width (thickness) of the contour lines.
- **Label Spacing**: Set the spacing between the contour line and the label.
- **Number of Decimals**: Set the number of decimals to show in the contour labels.
- **Label Font**: Select this button to specify the font settings for the contour labels.

Click the [Apply] button to view the changes in an active 2D or 3D Viewer.

### 6.4.5 3D Gridded Data

The following section describes the available style settings for 3D-Gridded data objects.

To access the style settings, right-click on the surface data object in the Data Explorer, and select Settings... from the pop-up menu. Then, in the Settings dialog, expand the Style node to view the style settings.

**Cells**
The Cells node allows you to specify style settings for the grid cells.

The Show Cell check box allows you to show/hide the grid cells in the 3D gridded data object. When the check box is selected, you can choose how to show the cells in the Color combo box in the Fill Settings frame. With the Specified option, select the adjacent color swatch and select the desired color to fill the cells. If you select Color by Attribute, you can color each cell according to a specified attribute, e.g., heads. Color by attribute settings can be defined by selecting the Color node, located under the Cells node.

For more information on the color by attribute feature, please refer to "Color By Attribute" section.

The Show only Active Zone check box allows you to show/hide inactive grid cells.

**Vertices, Lines**

For information on the settings available in the Vertices and Lines nodes, please refer the [Points \ Vertices](#) and [Lines](#) respectively.
Slice

Show Slice will control the display status in the 3D Viewer. Use Virtual Grid option allows you to use a coarsened version of the true 3d grid dimensions. This option is recommended when you have moderate to large size grids (exceeding a few hundred thousand cells). If you have a small grid then this option can be turned off. For more details, see Virtual Grid Settings.

Under Slice Settings, specify the desired Layer, Row, or Column Number.

Under Fill Settings, the ByAttribute option is default and recommended for most cases.

Show Legend check box at the bottom of the window will add a color legend to the current 3d view.

Show Wireframe will render with wireframe instead of filled in cells. The example below illustrates this:
In the Settings tree, under Slice -- Colors, you can access the color page where you can choose which attribute you want to render; in the case of Properties (or Recharge and Evapotranspiration) you can render by Zone or by the specified Attribute (eg. Kx, Recharge rate, etc.) For more information on the color by attribute feature, please refer to "Color By Attribute" section.

**Colormap**
Settings for the Colormap are identical to those explained above for Slice.

**Plot Color Map on Cross-Sections**

Color map has an additional **Slice Type**, which is "Cross Section" which is shown below:
After selecting this Slice Type, you need to provide a polyline data object that contains one or more polylines representing the cross-sections you want to render. Polylines can be imported from a shapefile or DXF file, or created manually. See Creating New Data Objects for more details.

Select this polyline data object from the tree, then click on the to insert this into the field as shown above.

Click Apply and the display will update with the appropriate cross-section lines. An example for two cross-sections is shown below.
In the Settings tree, under Colormap -- Colors, you can access the color page where you can choose which attribute you want to render; in the case of Properties (or Recharge and Evapotranspiration) you can render by Zone or by the specified Attribute (eg. Kx, Recharge rate, etc.). For more information on the color by attribute feature, please refer to “Color By Attribute” section.

Isolines
Settings for the Isolines are identical to those explained above for Slice.

Isolines also has an option to plot on a Cross-Section; see Colormap on Cross Section as described above.

Under Select Data, choose the attribute you want to use for calculating Isolines: For Properties, you can choose from Zones or Attributes (eg. Kx).

Additional Settings for Line Properties allow you to adjust the Line color, style, width. And, the number of contour lines, or the contour interval, and the starting value (minimum) by which contour intervals will be calculated. Settings in the Labels tab allow you to adjust the font size and color and the decimal format.

In the Settings tree, under Isolines -- Colors, you can access the color page where you can choose which attribute you want to render; in the case of Properties (or Recharge and Evapotranspiration) you can render by Zone or by the specified Attribute (eg. Kx, Recharge rate, etc.). For more information on the color by attribute feature, please refer to “Color By Attribute” section.
**Isosurfaces**

![Isosurface Node](image)

The IsoSurface node allow you to create and modify one or more isosurfaces from 3D gridded attribute data. An isosurface is a 3D planar surface defined by a constant parameter value in 3D space. Isosurfaces are typically used for demonstrating the spatial distribution of a selected parameter. For groundwater modeling purposes, isosurfaces are generally used for representing the spatial distribution of heads, drawdowns and concentrations.

**Creating an Isosurface**

To create an isosurface, follow the steps below:
From the Attribute Name combo box, select the attribute from which the isosurface is to be created.

Specify the attribute value in the Attribute Value field.

Select the color method from the Color box. The isosurface can be displayed as a solid color (Custom) or rendered by a specified attribute (ByAttribute).

Use the Visible check box to show/hide the isosurface.

Use the Show Border check box to display/hide a color map of the element value on the borders (sides) of the model domain when the isosurface intersects the edge of the model domain.

Use the Show in Cutaway check box to make the isosurface visible/invisible in cutaways.

Use the transparent check box to enable/disable transparency. If enabled, use the Transparency slider to set the level of transparency/opaqueness.

Click the [Add] button to create the isosurface.

The isosurface will be added to the isosurface table.

Modifying an Isosurface

To modify an existing isosurface, follow the steps below:

- Select the isosurface from the isosurface table.
- Make the modifications to the desired settings, e.g., attribute name, attribute value, color, etc.
- Click the [Change] button to apply the changes.

**Time**
The Time node provides a list of all the time steps in the 3D gridded data object, and allows you to select the desired time step data to display in the 3D Viewer window. For 3D gridded data objects generated by steady state flow models, only one time step will be available. For 3D gridded data objects generated by transient flow models, multiple time steps will be available (as defined in the Translation settings in VMOD Flex, i.e, Translation / Time Steps).
The ThreeSlices option allows you to see a slice through a selected model layer, row, and column at the same time in the 3D view. This is the display option that is used when you view 3D Gridded objects in the 3D View of the Flex Viewer. An example is shown below, with a slice through layer 2, and the centermost row and column in the Airport Transport tutorial project.
The generic options for "Slice" apply to the ThreeSlice including the Wireframe and "Use virtual grid" option.

The Colors and Legend options under ThreeSlice are also the same as described in earlier sections for Slice

The following options are unique to ThreeSlice:

- **Presentation Style**: Choose between Cells and Surface; when Surface is selected, you will see a color map with Isolines; under the Isoline node in the Settings tree, you can adjust the contour interval and line style.
- **Below Presentation Style**, select the desired Layer, Row, and Column number for the slice.

**VelocityMap**

*(Only available for output velocities)*
The VelocityMap settings are only available for output velocities and allow you to visualize output velocities in a variety of ways.

**General**

Groundwater velocities can be shown along a specified row, column, and/or layer as directed vectors along the plane with options for the arrow lengths:

- **Show**: if selected, velocities will be displayed using the options described below

- **Show normal velocity color map**: if selected, velocities perpendicular to the selected viewing planes (row, column, and layer pickers) will be displayed as a color map using the options on the next tab (see below).

- **Show isolines**: if selected and the normal velocity color map is also selected, isolines for the perpendicular velocities will be displayed using the intervals on the colormap options on the next tab (see below).

- **Color**:
  - **Custom**: velocity vectors will be displayed in the specified color selected on the right
  - **By normal velocity**: velocity vectors will be based on the flow perpendicular to the selected plane using the color options on the next tab (see below).

- **Velocity Type**:
  - **Darcy**: velocities are derived by interpolating inter-cell fluxes across the cell interfaces
  - **Average Linear**: average linear velocities (also called seepage velocities) are estimated by dividing the Darcy velocity by the effective porosity

- **Show Vectors as**:
  - **Direction**: View flow direction vectors (not-to-scale velocity vectors)
- **Projection**: View projections of the velocity vectors onto the current plane of view.
- **Magnitude**: View velocity vectors scaled according to the magnitude of the flow velocity in any direction.

- **Scale**:
  - **Linear**: velocity vectors are scaled by the linear value
  - **Logarithmic**: velocity vectors are scaled by the logarithmic value

- **Scale Factor**: velocity vectors are scaled by the specified value

**Color**

Color options for the velocity vectors apply to the velocities perpendicular to the displayed row, column, and/or layer selected on the General tab. Positive values (default red color ramp) correspond to the positive X, Y, and Z directions, typically east, north, and up,
respectively. Negative values (default blue color ramp) correspond to the negative X, Y, and Z directions, typically west, south, and down, respectively. Areas with velocities below the specified in-plane range threshold will be displayed using the specified in-plane color.

Legend
Legend options allow you to format the legend for the velocity colormap.
6.4.6 Settings Color Classifications

For those data objects that attributes that can be rendered, you will see a Colors page in the Style node of the Settings page. You can now adjust the values assigned to the colors to suit your needs (for example for Calculated Heads, Concentrations, Kx, etc.). A common application of this would be when you are viewing transport concentrations, and you want a value threshold to be assigned to a specific color (eg. 10 mg/l for a water quality).

Default Color Palette for Transient Data

After a model run, when you first view maps for transient data objects (heads, drawdown, concentration, water table), the color palette you see in the color legend will be a default. The color palette (including min/max values) will be re-calculated each time you change the output time. However, once you load the settings for the specific 2D or 3D view, then the values in the Color Classification will be applied globally for all output times.

Load the settings for the desired 2D or 3D view, and you should see the color page below (this example is for Concentration for the Layer (2D) View).
In the grid at the bottom of the window, you can adjust the value that are assigned the various colors. When using this option, you typically should set the Color Classification Type to be "Classified". Doing so, will assign a specific color to a range of discrete values. Enter the values working from highest to lowest (from top to bottom).

**Explanation of Color Values**

Using the data set and settings from the figure above, the color values were adjusted as follows.

In the example below, there are 4 color classes:

<table>
<thead>
<tr>
<th>Color</th>
<th>Value</th>
<th>Range of Values Assigned to this color</th>
</tr>
</thead>
<tbody>
<tr>
<td>Red</td>
<td>2000</td>
<td>&gt;1000 to the Maximum data value</td>
</tr>
<tr>
<td>Pale</td>
<td>1000</td>
<td>&gt;10 to 1000</td>
</tr>
<tr>
<td>Green</td>
<td>10</td>
<td>&gt;0.1 to 10</td>
</tr>
<tr>
<td>Aqua-Blue</td>
<td>0.1</td>
<td>Minimum data value to 0.1</td>
</tr>
</tbody>
</table>

2D viewer with "Surface" rendering on:
2D viewer with Cells rendering on.
As you change different output times, the user-defined color palette will be honored.
The settings need to be defined separately for the 2D and 3D View. Once you close a view (e.g. stand-alone 3D viewer), the color values will be lost, and you will need to re-define them.

In addition, in the case of rendering for MODFLOW/MT3D results (heads, drawdown, concentrations), the user-defined values will be lost upon re-translating and running the model, so you will need to re-define these values.
7 Conceptual Modeling Workflow

A Conceptual Model refers to a basic, high-level representation of the hydrogeological system being modeled. It will form the foundation for one or more numerical models. In Visual MODFLOW Flex, the conceptual model is completely grid and simulator independent. This means you define the inputs using your raw data objects (surfaces, polylines, polygons, etc.). The grid or mesh is only introduced at the time of launching a numerical model. This allows you to:

- Convert the conceptual model to multiple numerical models for uncertainty analysis
- Convert the conceptual model to a MODFLOW or FEFLOW model
- Easily update corresponding numerical models as your conceptualization changes

The sequential step in building the conceptual model are presented in the workflow navigator and are summarized below:

1. Define Modeling Objectives
2. Collect Data Objects
   (through importing or creating)
3. Define the Conceptual Model Area
4. Define Model Structure
   (vertical layering and resulting volume models)
5. Define Property Zones
   (parameter values for Conductivity, Initial Heads, Storativity)
6. Define Boundary Conditions
   (constant heads, drains, pumping wells, surface water network, etc)
7. Select Grid Type: Define Finite Difference Grid, Define Finite Element Mesh, or Define UnStructured Grid
8. Convert to Numerical Model
   (generate numerical inputs for a MODFLOW model)
9. Translate to Finite Element Model
   (generate FEFLOW ASCII .FEM file for running in FEFLOW)
10. Convert to Unstructured Model
    (generate numerical inputs for a MODFLOW USG model)
When a new conceptual model is created, a new Model Explorer tree is added to the Conceptual Model Explorer in the main Visual MODFLOW Flex window.

The Model Explorer tree consists of a fixed folder structure that is designed to guide you through the workflow of building your conceptual model.

---

**Before You Start!**

If you need to create, modify, or maintain a model that utilizes any of the following features, you must continue to use Visual MODFLOW Classic interface for this:

- Flow Engines (MODFLOW-96)
- Flow Packages (ETS1, MNW, STR)
- Transport Engines (MT3D99, PHT3D)
- MGO

---

### 7.1 Define Modeling Objectives

The first step in defining your conceptual model is to Define the Modeling Objectives, as shown below. Modeling Objectives are divided into two sections:

- **Flow Objectives**
- **Transport Objectives**
Visual MODFLOW Flex supports numerous groundwater flow models developed by the USGS and others. For the purposes of clarity and to distinguish them from particular regional and site-specific models, these models are referred to as "engines". Visual MODFLOW Flex currently supports the following engines:

### Engines supported in Visual MODFLOW Flex

<table>
<thead>
<tr>
<th>Flow</th>
<th>Transport</th>
<th>Utilities</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODFLOW-2000</td>
<td>MT3DMS</td>
<td>ZONEBUDGET</td>
</tr>
<tr>
<td>MODFLOW-2005</td>
<td>RT3D</td>
<td>MODPATH</td>
</tr>
<tr>
<td>MODFLOW-LGR</td>
<td></td>
<td>MOD-PATH3DU</td>
</tr>
<tr>
<td>MODFLOW-NWT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MODFLOW-USG¹</td>
<td></td>
<td>ZONEBUDGET-USG</td>
</tr>
<tr>
<td>MODFLOW-SURFACT²</td>
<td>MODFLOW-SURFACT²</td>
<td>MODPATH</td>
</tr>
<tr>
<td>SEAWAT³</td>
<td></td>
<td>ZONEBUDGET</td>
</tr>
<tr>
<td></td>
<td></td>
<td>MODPATH</td>
</tr>
</tbody>
</table>
Notes:
1 - MODFLOW-USG uses unstructured grids and is supported under a separate workflow process as compared to the standard numerical workflow.
2 - MODFLOW-SURFACT is a commercial groundwater modeling engine that supports coupled groundwater flow and transport simulations.
3 - SEAWAT is a coupled variable density groundwater flow and transport model.
4 - MOD-PATH3DU is a particle-tracking algorithm developed by a collaboration between S.S. Papadopulos & Associates (SSPA) and the University of Waterloo. It is not installed directly with Visual MODFLOW Flex and must be downloaded and installed separately from the mod-PATH3DU website.

Flow Objectives

The first step in the workflow for developing and running numerical models in Visual MODFLOW Flex is to define the modeling objectives. This entails selecting the desired flow and transport simulation options. The selected combination of flow options (e.g. saturated groundwater flow) and transport options (e.g. reaction and sorption models) will narrow the list of available flow and transport engines and generate associated input variables (including units and default values). The purpose of this step is to develop the necessary inputs required to build the model based on your understanding of the relevant physical processes present within the study area.

The flow engines supported in Visual MODFLOW Flex are presented in the table below, which provides a preview of what engines are available based on your selected flow and simulation types.

### Flow Engines and Modeling Objectives Supported in Visual MODFLOW Flex

<table>
<thead>
<tr>
<th>Flow Type</th>
<th>Saturated Flow</th>
<th>Variably Saturated Flow</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Engines</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MF-2000</td>
<td>![ ]</td>
<td>![ ]</td>
</tr>
<tr>
<td>MF-2005</td>
<td>![ ]</td>
<td>![ ]</td>
</tr>
<tr>
<td>MF-NWT</td>
<td>![ ]</td>
<td>![ ]</td>
</tr>
<tr>
<td>MF-LGR</td>
<td>![ ]</td>
<td>![ ]</td>
</tr>
<tr>
<td>MF-USG</td>
<td>![ ]</td>
<td>![ ]</td>
</tr>
<tr>
<td>SEAWAT</td>
<td>![ ]</td>
<td>![ ]</td>
</tr>
</tbody>
</table>

© 2019 by Waterloo Hydrogeologic
Notes:
1 - Abbreviations for variably saturated flow processes in the Vadose Zone:
   UZF = unsaturated zone flow package is based on 1-D unsaturated flow of a diffusion wave pseudo soil
   V = Van Genuchten relative permeability and saturation relationship
   BC = Brooks-Corey relative permeability and saturation relationship

Under Property Settings, you can modify the default flow parameters.

MODFLOW-LGR & MODFLOW-USG and Transport

The MODFLOW-LGR and MODFLOW-USG flow engines do not currently support contaminant transport with MT3DMS or RT3D.

- The Start Date of the model corresponds to the beginning of the simulation time period. It is important to define a relevant start date since your field measurements (observed heads and pumping schedules) will be defined with absolute (calendar) date measurements, and must lie within the simulation time period.
The start date will be used to retrieve pumping well and head/concentration observation data for the model run. When you define well data with absolute (calendar) dates, it is important that your start date reflects the actual start time for the model run. **Absolute time well data must fall on or after that start date.** Otherwise, these data will not be included in the simulation.

Also the start date cannot be changed once it has been set. If you inadvertently set the wrong start date, you can import your pumping well data and observation data in relative times (e.g. starting at 0), and you will see no difference in the numerical model inputs/outputs.

### Setting the Start Date

Visual MODFLOW Flex uses a standard Windows date picker; a few tips are shown below on how to use this. By default, the current date will be set as the Start Date but you are encouraged to set this to an appropriate date for your model simulation.

Click on the button shown below, to load the Windows date picker.
The standard Windows calendar will appear. Click on the month in the header (as shown below):

![January 2019 Calendar]

All months for the current year will appear as shown below. Click on the year in the header:
A range of years will then appear as shown below. Click on the range of years in the header:

A list of years for the previous decade will appear. You can then use the < or > buttons to change the year. Once you have reached the desired year (2000 for this example), select this on the calendar as shown below:

A list of months will then appear for that year. Select January for this example, as shown below.
Finally, select "1" from the calendar as shown below:

The selected date will then appear for the Start Date.

**Transport Objectives**

An explanation of the Transport Objectives is available in the section Numerical Model - Define Objectives.

Contaminant transport simulation can be enabled and setup in the conceptual model workflow. The species concentrations (for sinks/sources) can be defined while you define your conceptual boundary conditions. However, the transport properties (initial concentrations, dispersion, etc.), are defined in the Numerical Model workflow, after you have defined a grid, and completed the "Convert to Numerical Model" step. This is explained in the section Define Properties (Numerical Model).

- Click (Next Step) to proceed.
7.2 Collect Data Objects

At this step in the workflow, you collect the data you want to use to build and interpret your conceptual model.

The conceptual modeling workflow is data driven. This means that you first need to "collect" the appropriate data objects in order to use these at a particular step in the workflow. Data objects can be "collected" through several means:

- **Importing Data**: Import GIS data (shapefiles, CAD files), Gridded data, images, points/wells in excel spreadsheets, or XYZ points in text format.
- **Creating Data Objects**: digitizing new point, polygon, or polyline data objects.
- **Creating Surfaces**: Interpolate XYZ points using Krigging, Natural Neighbor, or Inverse Distance. The resulting surfaces can be used to define geological layers or 2D parameter distributions (Kx, Recharge, etc.)
Minimum Data Requirements

In order to build your conceptual model, you require at least the following data objects:

- Two Surfaces: One for the top and one for the bottom of a geological unit
- A polygon that represents the model area

Once you have the data loaded, Click (Next Step) to proceed.

You may return to this step in the workflow at any time during the model process to import or create new data objects.

See the following table for some typical data object types and how they are used in the conceptual model workflow.

<table>
<thead>
<tr>
<th>If you have...</th>
<th>First you should...</th>
<th>Then you can..</th>
</tr>
</thead>
<tbody>
<tr>
<td>River locations in a shapefile</td>
<td>Import these as Polyline data objects</td>
<td>Select this data object when creating a river boundary condition</td>
</tr>
<tr>
<td>Geological layers in a Surfer .GRD or ASCII GRD</td>
<td>Import these as Surface data objects</td>
<td>Select these data objects when defining horizons</td>
</tr>
<tr>
<td>Air photo with river location</td>
<td>Import this as a Map image</td>
<td>Select this new polyline data object when creating a river boundary condition</td>
</tr>
<tr>
<td></td>
<td>Then Create a new Polyline data object</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Digitize the polyline in a 2D Viewer</td>
<td></td>
</tr>
<tr>
<td>XYZ points for geological contacts</td>
<td>Import these as Points data objects</td>
<td>Select these surfaces when defining horizons or numerical model layers</td>
</tr>
<tr>
<td></td>
<td>Then &quot;Create Surfaces&quot; from these data objects</td>
<td></td>
</tr>
<tr>
<td>Raster Grid of Kx or Recharge data (from Surfer, ESRI .GRD)</td>
<td>Import these as surface data objects</td>
<td>Select these data objects when defining properties or Recharge boundary conditions</td>
</tr>
</tbody>
</table>
7.3 Create New Conceptual Model

At this step, provide the conceptual model area and the start date.

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Provide a polygon for the conceptual model area</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites</td>
<td>A polygon data object has been imported or created</td>
</tr>
<tr>
<td>Result</td>
<td>Model Explorer tree is defined with the Model Boundary</td>
</tr>
<tr>
<td>Next Steps</td>
<td>Define Model Structure</td>
</tr>
</tbody>
</table>

Once you have imported sufficient raw data into your project, you can begin to construct one or more conceptual models using imported or digitized data objects as building blocks. At this step, you need to define some basic information about the conceptual model, and provide a polygon that represents the model area.
You can import a polygon shapefile/DXF file, as described in the section Import Polygons. Or, if you do not have a polygon, you can create an empty polygon data object, then digitize the geometry. Please refer to the section Creating New Data Objects for more details.

In the Define Conceptual Model window (as shown above), define the settings for the conceptual model.

- Enter a unique name for the conceptual model in the Name field.
- Enter a description of the conceptual model in the Description field (optional).
- Specify a start date from the Start Date combo box. This will be used for calculating the start date for transient model runs.

**Please Note:** If you are using pumping wells, the start date must match the first start time for your pumping schedule.

- From the Data Explorer, select the polygon data object that represents the conceptual model horizontal boundary, and then click the button.

**Please Note:** The model area must be defined using a simple polygon. It cannot be defined using a complex polygon or one that contains multiple polygons. A complex polygon is a polygon that intersects with itself and/or contains holes.

- Click the [Save] button.

Once you are finished, click (Next Step) to proceed.

**Model Explorer Tree**

Once a conceptual model is created, a new Conceptual Model tree is added to the Model Explorer. The conceptual model tree sets up the workflow for structural and property modeling, assigning boundary conditions, numerical grid creation, and numerical model translation. A typical conceptual model tree is shown below:
The Model Boundary node allows you to show/hide the conceptual model boundary in a 2D or 3D Viewer.

The Structure folder allows you to define the horizons and structural zones of the conceptual model. For more information on structural modeling, please see "Defining the Structure".

The Properties node allows you to define property zones for the conceptual model. For more information on property modeling, please see "Defining Property Zones".
7.4 Defining the Structure

At this step, provide the geological surfaces that will represent the tops and bottoms of the geological model.

Quick Overview

<table>
<thead>
<tr>
<th>Instructions:</th>
<th>Define geological surfaces for your conceptual model and provide Horizon hierarchy rules</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites:</td>
<td>Two or more surface data objects have been created or imported.</td>
</tr>
<tr>
<td>Result:</td>
<td>Horizons and Structural Zones are created</td>
</tr>
<tr>
<td>Next Steps:</td>
<td>Define Property Zones</td>
</tr>
</tbody>
</table>

At this step, you define the geological structure for the conceptual model, by converting surfaces to Horizons.

Horizons are stratigraphic layers (2D surfaces with topography) that define the upper and lower boundaries of the structural zones in a conceptual model. In Visual MODFLOW Flex,
horizons are created by clipping or extending interpolated surface data objects to the boundary of the conceptual model.

When horizons are created, Visual MODFLOW Flex will automatically generate the Structural Zones (geologic formations) between the horizons, which can be used later to define property zones.

This segment is separated into the following sections:

- Defining Horizons
- Explanation of Horizon Types
- View Structural Zones

Defining Horizons

Before You Start!

Make sure you have all the surface data objects that you want to use to construct the geological model. Surfaces can be created or imported as explained at the Collect Data Objects step. Also, it is currently not possible to change the Horizons after the model structure has been finalized.

To create a new horizon:

- From the Data Explorer, select the surface data object that will be used to generate the horizon.
- Click the Blue Arrow button to insert it into the Horizon Information table. If the selected surface is greater than the conceptual model area, it will be clipped by the conceptual model boundary polygon. If the surface is less than the conceptual model area, it will be extended to the conceptual model boundary.

Please Note: Surfaces should be added from top to bottom, starting with the top surface (ground surface) and ending with the bottommost surface.

- Click the Add Horizon button to add a new horizon row to the Horizon Information table.
- In the Name column, type in a unique name for the horizon.
In the Type column, select the appropriate horizon type from the combo box. For information on each horizon type, please refer to "Horizon Types".

Repeat the steps above to add additional horizons. Remember you must have at least two horizons before Visual MODFLOW Flex can create the structural zones.

You can preview the horizons in 3D Viewer, by clicking the [Apply] button.

Finally, click the [ ] button to create the horizons and proceed to the next step.

Once created, the horizons will be added to the Model Explorer Tree under the Horizons node (shown below).
Horizon Types
Each horizon can be assigned a particular type, which defines the relationship to other horizons in the conceptual model. This prevents intersecting layers and establishes layers that satisfy both FEFLOW and MODFLOW requirements. Each horizon type is described below.

Erosional horizons can be used as the highest or as an intermediate horizon, but not as the bottom of the conceptual model. This type of horizon will truncate all horizons below it, including the base horizon.

Base horizons can be used as the lowest horizon in the conceptual model. Any conformable horizon types will lap onto it, while all erosional or discontinuity horizons will truncate it.

Discontinuity horizons represent an erosional surface in the middle of a stack of horizons. It can never be the highest or lowest horizon. Horizons above it up to the next discontinuity or erosional horizon will lap onto it, while all horizons below it will be truncated by it. These horizons can be thought of as the top or base of a sequence.

Conformable (default) horizons will be truncated by erosional, base and discontinuous horizons. Lower conformable horizons will be truncated by upper conformable horizons. If a conformable horizon is above an erosional horizon, the conformable horizon will “conform” to the erosional horizon (it will be pushed up by the erosional horizon).
The horizon rules described above are applied after all the horizons are calculated. If one of the horizons will be truncated by an erosional, base, or discontinuity horizon, it is a good idea to extend the input data beyond these unconformable horizons in order to truncate them properly.

**Demonstration of Horizon Types**
The image below shows three surfaces in a 3D Viewer. The surfaces are colored Red, Green, Blue, from top to bottom, respectively.

You will see that there are spots where the green surface intersects with the red surfaces, and likewise, where the blue surface intersects with the green surface. For numerical models (MODFLOW and FEFLOW), this geometry is not permitted (as it will lead to zones with negative thickness), which is why surfaces are converted to horizons. Assigning a horizon type will eliminate the intersections.

When horizons are generated from these surfaces, and each surface is set to Conformable by default. Conformable horizons are truncated such that each one conforms to the horizon above it, as shown in the following image.
If the middle horizon is set to Erosional, with the top and bottom set to Conformable, the topmost surface (red) is pushed up, as shown in the following image.
View Structural Zones

During the horizon creation process, Visual MODFLOW Flex automatically generates the structural zones between the defined horizons within the horizontal extent of the conceptual model boundary. To view the generated structural zones, in the Model Explorer Tree, expand the Structure Node and then expand the Zones node.
Zones are given a default name, e.g., Zone1, Zone2, Zone3 etc., which cannot be modified.

You can view the zones in an active 3D Viewer window by checking the empty check box beside the zone name. For more information on data viewers, please refer to "Data Viewers" section.

7.5 Defining Property Zones

At this step, define the flow property values for the geological formations.

Quick Overview

<table>
<thead>
<tr>
<th>Instructions:</th>
<th>Define flow properties for the conceptual model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites:</td>
<td>Structural Zones are created.</td>
</tr>
<tr>
<td></td>
<td>If you wish to use a spatially-distributed property values, you have imported or created surfaces.</td>
</tr>
<tr>
<td></td>
<td>If you wish to subdivide zones into lateral regions, you have imported polygon shapefile(s) or digitized polygons.</td>
</tr>
<tr>
<td>Result:</td>
<td>One or more property zones are created</td>
</tr>
<tr>
<td>Next Steps:</td>
<td>Define Boundary Conditions</td>
</tr>
</tbody>
</table>
At this step, you define Property Zones for the conceptual model. Each property zone is assigned appropriate property attributes, e.g., conductivity, storage, and initial heads.

A groundwater flow model requires many different types of data to simulate the hydrogeological processes influencing the flow of groundwater. In Visual MODFLOW Flex, the hydrogeological characteristics of the model are classified into the following parameter groups:

- **Conductivity**: horizontal conductivity in the horizontal/x-direction (Kx), in the horizontal/y-direction (Ky) and in the vertical/z-direction (Kz)
- **Storage**: specific storage (Ss), specific yield (Sy), effective porosity (Ep), total porosity (Tp)
- **Initial Heads**

By default, Visual MODFLOW Flex automatically assigns the default property parameter values, specified in the Project Settings (see "Project Settings" for more details) to the entire model domain. However, in most situations, the flow properties will not be uniform throughout the entire model domain, and it will be necessary to assign different property values to different areas of the conceptual model. This can be accomplished by creating Property Zones. In Visual MODFLOW Flex, a property zone is a specified 3D-volume, generated from structural zones, with user-defined hydrogeologic attributes.

Property zone geometry can be defined using one or more existing structural zones. As such, property zones can only be generated after horizons have been defined in the conceptual model. Please see Defining the Structure for more details.

Visual MODFLOW Flex supports various methods for assigning values to hydrogeologic parameters. The method used for defining attributes can be defined on the parameter level, allowing you to use different methods for different parameters. The supported methods include:

- Use Constant Value
- Use Surface Data Object
- Use 3D Gridded Data Object
- Use Shapefile

The following sections provide information on the following topics:

- Defining a New Property Zone
- Assigning Property Parameters
- Editing Property Zones
- Deleting a Property Zone
Creating New Property Zone

Before You Start!

Make sure you have imported or created all the data objects that you plan on using for properties. If you wish to use a property distribution, Import Surfaces or Create Surfaces.
If you wish to use shapefiles, Import Polygons or Digitize Polygons.

Before you can create a property zone, you must have already defined horizons for the conceptual model. For more information, see Defining the Structure.

To define a new property zone, follow the steps below:

- Enter a Name and Description (optional) for the property zone in the Name and Description fields, respectively.
- Select the method by which the property zone geometry will be defined. There are two options: Use Structural Zone(s) and Use Polygon Data Object.

Using Structural Zone(s)
This method allows you to create a property zone from existing structural zones in your conceptual model, i.e., zones generated from horizons.
Click on the [Use Structural Zone] button as shown above.
Select a zone from the Model Explorer tree (under the Zones node), and then click the button to insert the zone in the Structural Zones field.
Click the Add button to add and combine structural zones.
Apply to all... button allows you to apply property attribute values for all polygons at once.
Proceed to Defining Property Zones

**Using Polygon Data Object**
This method allows you to define a property zone using both a structural zone and a polygon data object. The polygon data object is used to define the horizontal extent of the property zone and therefore must be fully contained within the conceptual model boundary. The structural zone is used to define the volume, i.e., the vertical extent of the property zone.
• Select a polygon data object from the Data Explorer, and click the button to insert the data object in the Select Polygon Data field.

  **Please Note:** The selected polygon cannot contain multiple parts, overlapping shapes or holes. These features are currently not supported for property zone creation. If your polygon does not meet this criteria, it can be edited using the 2D Viewer editing tools. For more information on this topic, please see "Creating New Data Objects"

• Next, select a structural zone from the Model Explorer tree, and click the button to insert the data object in the Define Volume frame.

• Apply to all... button allows you to apply property attribute values for all polygons at once.

**Assigning Property Parameters**

Once the geometry has been defined, you can assign parameter values to the property zone.

• Select the group of parameters that will be defined, e.g., conductivity, storage or initial heads. The data input grid below will display the appropriate parameters based on which parameter group is selected. For example, if conductivity is selected, the data input grid will show the parameters Kx, Ky, and Kz. The data input grid will already be populated with the default values specified in the Project Settings (File > Project Settings... ).
Visual MODFLOW Flex provides various methods for assigning parameter attributes. The available methods include: Constant Value, Use Surface, Use 3D Gridded Data and Use Shapefile (available only when property zone is defined using polygon data object). The type of method used can be specified per parameter. For each parameter in the data input grid there is a combo box in the Method row (shown below).

Each method is described in the following sections.

**Constant Value**
The Constant Value method is selected by default for each parameter in the data input grid and allows you to specify a spatially constant value for the parameter. If you do not wish to use the default value, enter a new value.

**Use Surface**
The Use Surface method allows you use an existing surface data object to define spatially-variable attribute values. This is ideal if you have a Surfer .GRD or ESRI ASCII GRD file containing parameter values. Follow the steps below:

- Select "Surface" as the method from the combo box. You should then see the $\rightarrow$ button become activated under the "Value" column
- From the Data Explorer, select the desired surface data object
- Click the $\rightarrow$ button to insert this data object into the Object field.

⚠️ Please Note: The selected surface data object must cover the entire area of the property zone, or else the data object cannot be used.

**Use 3D Gridded Data**
The Use 3D Gridded Data method allows you to use an existing 3D gridded data object to define spatially-variable attribute values. Follow the steps below:

- Select "3D Grid" as the method from the combo box. You should then see the $\rightarrow$ button become activated.
- From the Data Explorer, select the 3D Gridded data object
- Click the $\rightarrow$ button to insert it into the parameter field.
- Select the desired attribute value from the "Mapping" combo box.

**Please Note:** The specified 3D Gridded data object must horizontally and vertically overlap the defined property zone geometry, or else the data object cannot be used.

**Use Shapefile Attribute**

The Use Shapefile method is only available when you define the property zone geometry using a polygon data object. This method allows you to assign an attribute value using an attribute from the specified polygon data object. Follow the steps below:

- If you have not already done so, create the Property Zone [Using Polygon Data Object](#), as described above.
- Select the Use Shapefile Attribute from the method combo box.
- Under Attribute column, the combo box contains all the attributes of the specified polygon.
- Select the desired attribute from the combo box.

Once the property zone attributes have been defined, click the [Save] button to create the property zone. Once created, the property zone is added to the Model Explorer tree under the Properties node and under the appropriate parameter category node.

**Transport Property Zones**

If you need to include contaminant transport in your model, then the properties for transport will be done in the numerical workflow. Be sure that in the Define Modeling objectives step (in the Conceptual Model), that you have selected “Transport Active”. After you define a numerical grid, and create a numerical model, the transport properties will be defined at the “Define Properties” step.
7.6 Defining Boundary Conditions

At this step, define the boundary conditions for the conceptual model.

Quick Overview

Instructions: Define hydrological boundaries (sources/sinks) for the model.

Pre-requisites:
- Structural Zones have been created.
- Property Zones have been created.
- You have imported and/or created the data objects you want to use for defining boundary conditions. For example, polylines for the geometry or Surfaces or time schedule for defining the attributes.

Result: A "conceptual" boundary condition object is created

Next Steps:
- Create Finite Difference Grid,
- Create Unstructured Grid, or
Background

Every conceptual model requires an appropriate set of boundary conditions to represent the system’s relationship with the surrounding systems. In the case of groundwater flow model, boundary conditions will describe the exchange of flow between the model and the external system.

- Specified Head
- River
- General Head
- Drain
- Recharge
- Evapotranspiration
- Lake
- Specified Flux
- Wall (Horizontal Flow Barrier)
- Pumping Well

The workflow for defining a pumping well boundary conditions and a Wall (Horizontal Flow Barrier) is different than that of defining other boundary conditions such as recharge, specified head, river, etc. Please see the following section for information on defining pumping well boundary conditions. For information on how to define all other types of boundary conditions, please skip to "Defining Other Boundary Conditions" section.

Defining Boundary Conditions

In Visual MODFLOW Flex, the boundary condition types are separated into the following categories:

- Boundary Conditions (standard) (Constant Head, Rivers, Drains, General Head, Recharge, EVT, etc.)
- Wall (Horizontal Flow Barrier)
- Pumping Wells

Modifying Boundary Conditions

From the Model Explorer tree, right-click on the desired boundary condition, and select Edit Boundary Condition... from the pop-up menu.

The Edit Boundary Condition dialog box will display on your screen, allowing you modify the input parameters for the boundary condition. For more information on defining parameter attributes, please refer to "Boundary Conditions Overview" section.

Once modifications have been made to the input parameters, click the [Finish] button to save the changes.
Deleting Boundary Conditions
To delete a boundary condition, follow the steps below:

- From the Model Explorer tree, right-click on the desired boundary condition and select Delete from the pop-up menu.
- You will be prompted with a confirmation message. Click the [Yes] button to delete the boundary condition.

**Warning!**
Please be aware that there is no undo function to recover a deleted boundary condition. Please exercise caution when deleting boundary condition.

Parameters for Transport Boundaries (Sinks/Sources)
If you need to include contaminant transport in your model, then the species concentrations for transport sinks/sources must also be defined while you create your conceptual boundary conditions. By default, species concentrations will be undefined when you create a new boundary condition. An undefined value is identified by the value “-1” for the species concentration (eg. Conc001, Conc002). If you wish to assign a contaminant mass to a specific boundary condition object, then replace the default -1 value with a representative contaminant concentration.

Limitations
In Visual MODFLOW Flex you can create boundary conditions in the conceptual model and in the numerical model. When you edit a conceptual boundary condition object, its corresponding numerical representation will not get updated; likewise, when you update a numerical boundary condition, its corresponding conceptual representation will not get updated. The one exception to this rule is Pumping Wells.

7.6.1 Define Boundary Conditions (Lines/Polygons)

**Before You Start!**

Make sure you have imported or created all the data objects that you want to use for Defining Boundary Conditions. Refer to the Collect Data Objects step.
Define Geometry

The first step involves selecting the boundary condition type and specifying the location of the boundary condition on the simulation domain.

- From the Select Boundary Condition Type combo box, select the desired boundary condition type. For more information on each boundary condition type including the data requirements for MODFLOW, please see “Boundary Conditions Overview”
- Enter a Name and a Description (optional) for the boundary condition. The specified name will appear in the Model Explorer tree, once the boundary condition is created.
- Next, select where to apply the boundary condition on the simulation domain by selecting an option from the Where to apply on the Simulation Model Domain combo box. The type of options available in this combo box depend on which boundary condition type is selected. The table below summarizes the available options for each boundary condition type:

![Define Boundary Condition Dialog Box](image)

<table>
<thead>
<tr>
<th>Top</th>
<th>Bottom</th>
<th>Side</th>
<th>Intermediate</th>
</tr>
</thead>
</table>

© 2019 by Waterloo Hydrogeologic
Boundary conditions can be applied to the top, bottom, intermediate layers or sides of the simulation model domain, using imported or digitized Polygon or Polyline data objects, or by manually selecting the sides of the simulation domain using an interactive 3D Viewer.

Each option is described below:

- **Top**: Selecting this option will apply the boundary condition to the top layer of the simulation domain.
- **Bottom**: Selecting this option will apply the boundary condition to the bottom layer of the simulation domain.
- **Side**: Selecting this option will allow you to apply the boundary condition to a single side or combination of sides of the simulation domain. When this option is selected, you will be required to define the geometry of the boundary condition by selecting the desired sides of the simulation domain using an interactive 3D Viewer. For more information on how to do this, please see "Defining Boundary Condition Geometry (Sides)" section.
- **Intermediate**: Selecting this option will allow you to assign the boundary condition to an intermediate model layer within the simulation domain, by specifying a Connection Elevation value. Upon translation, Visual MODFLOW Flex will assign the boundary condition to the appropriate model layer based on the defined connection elevation value.

The next step is to define the geometry of the boundary condition. Boundary conditions that are applied to the top, bottom or intermediate layers can be defined by using a Polygon (areal) or Polyline (linear) data object from the Data Explorer.

- From the Data Explorer, select the polygon or polyline data object that represents the geometry of the boundary condition.
- Click the button to insert the data object into the Select a polyline or polygon from the Data Explorer field.
Please Note: Some boundary conditions only support one type of geometry, either polygon or polyline.

Polygons and polyline data objects that extend beyond the model domain must be “clipped” before they can be used to define boundary conditions. Polygons and polylines can be clipped using the “clip to polygon” data operations, which can be accessed in the data object settings (right-click on the polygon or polyline in the Data Explorer, and select Settings... from the pop-up menu). For more information on data object operations, please see “Performing Operations on Data” section.

- Click the [Next] button to proceed to the next step.

For boundary conditions that are to be applied to the Side(s) of the simulation domain, Visual MODFLOW Flex allows you to define the geometry by selecting the appropriate side(s) using an interactive 3D Viewer window. This process is described in "Defining Boundary Condition Geometry (Sides)" section.

If you select a polygon data object to define the geometry of the boundary condition, the next step is defining boundary condition parameters. Please skip to "Defining Boundary Condition Parameters" section for information on this topic.

If you select a polyline data object to define the geometry of the boundary condition, the next step is defining zones for the selected polyline. This procedure is described in the following section.

Define Attributes

Visual MODFLOW Flex provides various methods for assigning parameter values to boundary conditions. Each parameter in the boundary condition can be set to Constant or Transient, and values can be assigned using attributes from various imported data objects. The available methods for assigning attributes include:

Parameter values must be entered in the units defined in the project settings. To read more about the project settings, please see "Units" section in the project settings.

Once the geometry has been defined (see previous sections), the next step is to define the boundary condition parameters. Although each boundary condition type requires a different set of parameters, the data input windows each have similar features and functionality. For information on required parameters for each boundary condition, please see the appropriate heading under "Boundary Conditions Overview" section.

A typical boundary condition data input window is shown below:
Select the Method for Defining Attributes (Polylines Only)

For polylines, there are two ways in which you can assign attributes to the boundary condition geometry:

- Define for the entire zone (default): This option allows you to assign boundary condition data to the entire zone.
- Define values at vertices: This option allows you to assign boundary condition data to the vertices along the zone (line), and then during translation, linear interpolation is used to determine the parameters for the cells that fall between the specified vertices. With this method, there are two options:
  - Define Start and End Points: This option allows you to define the attribute value only at the first and last vertex of a zone.
  - All Vertices: This option allows you to define attribute values for each vertex in a zone (first, last and all intermediate vertices).

Specifying Boundary Condition Data

Boundary condition data can be specified by using the Data Input Grid. Each column in the data input grid represents a required attribute for the selected boundary condition. For more information on the required parameters for each boundary condition, please see the appropriate section under "Boundary Conditions Overview"
Attribute data must be defined for feature (polygon or polyline) in the boundary condition object. Simply select the feature for which attributes are to be defined from the Feature List (the selected line or polygon will be highlighted yellow in the adjacent 2D Viewer preview), and then define the attribute data in the Data Entry Grid. Repeat this process for other features in the selected data object.

**Please Note:** Attribute data must be defined for all features (polylines or polygons) in the selected data object, in order to complete the creation of the boundary condition.

- [Apply to All]: use this option (located in the top middle portion of the window) to apply the current parameter values/methods, or mapping for the selected polyline (or polygon) to all other polylines (or polygons) in the selected data object. Useful when you have 10's or 100's of features in one data object, and they all reference a common data source (eg. a surface) or a constant value.

For polylines only, it is possible to define attributes at line vertices. In this case, the Points List will become available, where you can select the vertices that comprise the selected zone. For more information on assigning attributes to points, see "Select the Method for Defining Attributes (Polylines Only)" section.

For each attribute in the Data Input Grid, there are two combo boxes.
The combo box in the first row allows you to set an attribute as Static (Steady-State) or Transient (conditions change over time). When one or more parameters are set to Transient, the Transient Data button will become active. When selected, the Transient Data window will launch (shown below).

The Transient Data dialog allows you to define the stress periods and values for all the attributes in the boundary condition that have been set as “Transient”.

- Click the Add Row button to add a new row to the table.
- Enter a Start and End time, and a Value for each transient attribute.
- Press the [Enter] button on your keyboard.
• Repeat for additional stress periods.
• Click [OK] to save the transient data.

The second combo box provides different methods for assigning attribute values to the boundary condition.

The contents shown in this combo box depend on the attribute type, i.e., not all methods are available for every attribute. The available methods may include:

- Constant Value
- Use Surface
- Use Shapefile
- Use 3D Gridded Data
- Use Time Schedule (Transient only)

Each method is described below:

**Constant Value**

The constant value method allows you to define a single value for the entire zone. Upon translation, each grid cell comprising the boundary condition zone will be assigned the specified constant value.

When this method is selected (default), simply enter the desired attribute value in the Data Entry Grid.

ℹ️ **Please Note:** The values for each constant value attribute should be entered in the same units as defined in the Project Settings.

**From Surface**

This method allows you to define boundary condition attributes using an existing Surface data object. Upon translation, attribute values are calculated from the specified surface data object. A surface data object can be useful for defining an elevation attribute, i.e., River Stage, Constant Head, Lakebed Bottom etc.

ℹ️ **Please Note:** The surface data object must cover the entire conceptual model domain area.

When this method is selected, click the [From Surface] button to launch the Static Data Control dialog (shown below).
To specify a surface data object,

- Select the desired surface from the Data Explorer.
- Click the button to insert the surface data object into the attribute field.
- Repeat for other attributes that have been assigned this method.
- Click the [OK] button.

**From Shapefile**

The method allows you to use Shapefile attributes for defining boundary condition attributes (for example River stage, bottom, riverbed thickness, river width and riverbed conductivity or Constant Head starting head and ending head).

- Select the "Use Shapefile" method from the combo box
- Click the Use Shapefile button to launch the Shapefile dialog; as shown below:

  - The combo box contains all the attributes of the specified polygon used to define the horizontal geometry of the boundary condition.
  - Select the desired attribute from the combo box, and then click the [OK] button to close the dialog box.
From 3D Gridded Data
This method allows you to use spatially-variable attributes from a 3D Gridded data object for defining a boundary condition attribute. When this method is selected, the Use 3D Gridded Data button will become active. When selected, the 3D Gridded Data dialog will launch.

- From the Data Explorer, select the desired 3D Gridded data object
- Click the button to insert it into the Select 3D Gridded Data Object field.
- Once selected, the data object’s attributes are listed in the combo box below.
- Select the desired attribute from the combo box, and then click the [OK] button to close the dialog box.

⚠️ Please Note: The specified 3D Gridded data object must horizontally and vertically overlap the defined property zone geometry, or else the data object cannot be used.

Use Time Schedule
This method allows you to use a time-schedule data object for defining the stress periods and values of a transient attribute. When this method is selected, click the Use Time Schedule button to launch the Time Schedule dialog.

- From the Data Explorer, select the desired time schedule data object
- Click the button to insert it into the Select Time Schedule Object field.
- Once selected, the data objects attributes are listed in the combo box below.
- Select the desired attribute value from the combo box, and then click the [OK] button to close the dialog box.

Use Default Leakance
The Use Default Leakance option is used to calculated the leakance value for River, Drain, Lake and General Head boundary conditions using a mathematical expression containing
array variables (see the section “Using Mathematical Formulas and Array Variables in the Visual MODFLOW Flex User’s Manual for more information). If the Use Default Leakance option is selected, the leakance value will be calculated using a default formula associated with each boundary condition type. If this option is not selected, a leakance value will need to be entered manually.

The advantage of using the default leakance formula to calculate the leakance value for the group of grid cells is that each grid cell will be assigned a leakance value proportional to the size of the grid cell.

**Linear Interpolation Explained**

When the Define values at vertices option is selected, two sets of boundary condition data are required; one set for the grid cell at the Start Point of the line (or line segment), and one set for the grid cell at the End Point of the line (or line segment). The boundary condition data for the grid cells between the Start Point grid cell and the End Point grid cell will be linearly interpolated between these two points using the formula below:

\[ X_i = x_{SP} + \left( x_{SP} - x_{EP} \right) \frac{TVAR_{1,i}}{LENGTH} \]

where:

- \( X_i \) is the boundary condition parameter value at the ith grid cell along the line
- \( x_{SP} \) is the boundary condition parameter value at the Start Point of the line
- \( x_{EP} \) is the boundary condition parameter value at the End Point of the line
- \( TVAR_{1,i} \) is the cumulative length of the line at the ith grid cell along the line, as measured from the center of the Start Point grid cell through the center of each successive grid cell along the line (see following figure).
- \( LENGTH \) is the total length of the line, as measured from the center of the Start Point grid cell through to the center of the End Point grid cell (see following figure).

When the line is digitized from the Start Point to the End Point, each grid cell is numbered in sequence according to the order in which the line passes through each cell. If the line passes through the same grid cell twice, the grid cell will be numbered twice as seen for grid cell “4” and “6” in the following figure. As a result, the parameter value calculated for “grid cell #6” will over-write the parameter value calculated for “grid cell #4”.
7.6.2 Define Boundary Conditions to Sides

If you choose to apply the boundary condition to the Sides of the simulation domain, click the [Next] button to select which side(s) to apply the boundary condition to.
Applying boundary conditions to the sides of the simulation domain is accomplished by manually selecting the desired sides using an interactive 3D Viewer window.

- Click the [Show] button to display the interactive 3D Viewer.

The interactive 3D Viewer behaves just like any other 3D Viewer. You can zoom in and out, rotate and move the displayed simulation domain using your mouse. You can also change the color of the background, show/hide the axis, and change the vertical exaggeration.

**Creating a New Zone**

To create a new zone, follow the steps below:

- Click the [Create New Zone] button. A new row will be added to the Zones table. Here you can change the zone Name and Description, as desired.
- Click the [Start Selection] button. A new combo box called Selector will be added to the bottom of the interactive 3D Viewer (indicated below).

From the Selector combo box, select one of the following options:
- **Global**: Select all sides around the entire simulation domain
- **Horizontal**: Select the area(s) between two horizons, around the entire simulation domain.
- **Vertical**: Select an area(s) between two edges, spanning the entire height of the simulation domain.
- **Facets**: Select the area(s) that are bounded on the sides by edges of the simulation domain, and bounded at the top/bottom by horizons
Using your mouse, click on the simulation domain in the 3D Viewer, and select the appropriate sides for the boundary condition. When a side is selected, it will become highlighted.

**Tip!** You can select multiple verticals, sides, or facets by holding down the [CTRL] key on your keyboard while adding to your selection using left click(s) of your mouse.
Once the desired areas have been highlighted, click the [Done Selection] button to save the selections. Repeat the steps above to create additional zones.

When you are finished, click Next to define the attributes; for more details, see Define Boundary Conditions (Lines/Polygons)

7.6.3 Define Pumping Wells

At this step, you can define well boundary conditions.
Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Select a wells data object to be used as wells in the conceptual model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites</td>
<td><strong>Boundary Conditions are created</strong></td>
</tr>
<tr>
<td></td>
<td>You have imported a wells data object, with screens and pumping schedule information.</td>
</tr>
<tr>
<td>Result</td>
<td>A Conceptual Wells Boundary Condition object is created</td>
</tr>
<tr>
<td>Next Steps</td>
<td>• <a href="#">Create Finite Difference Grid</a>;</td>
</tr>
<tr>
<td></td>
<td>• <a href="#">Create Unstructured Grid</a>; or</td>
</tr>
<tr>
<td></td>
<td>• <a href="#">Create Finite Element Mesh</a></td>
</tr>
</tbody>
</table>

💡 Before You Start!

Make sure you have imported a Wells Data object that contains well location, screens, and pumping schedule. Refer to [Import Wells](#) for more details.

To add a new pumping well boundary condition, follow the steps below:

- Click on the Define Pumping Wells button; the following window will appear:
Select a pumping wells data object from the Data Explorer

Click the button to insert the data object into the Select Wells Data Object field. The wells should then show in the table.

Each pumping well must satisfy the following requirements in order to be deemed valid:

- The pumping well must located within the simulation domain.
- A screen must be defined for the pumping well
- A pumping schedule must be defined for the pumping.

Wells that do not meet these requirements will not be included. For information on defining well data, i.e., screens, pumping schedules, please see "Well Table" section

Visual MODFLOW Flex will then add the boundary condition under the Boundary Condition node in the Model Explorer tree.
**Changes to Wells in v.2015.1**

VMOD Flex v.2015.1 utilizes a new conceptual approach to pumping wells; if you wish to utilize this format, you will need to convert your wells to this format. For more details on this process, and the limitations, please see [Converting "Old" Wells to "New" Wells Format](#).

### Calculation of Well Rates

Visual MODFLOW Flex supports wells with multiple well screens throughout the depth of the well-bore, and well screens that partially penetrate a model layer. However, it is important to note that the MODFLOW WEL package does not have any special considerations for multiple well screens or grid cells containing partially penetrating well screens. MODFLOW treats wells as flux boundary conditions, such that each grid cell intersecting a well screen is assigned a specified flux. In a situation where a well is screened across several model layers, Visual MODFLOW Flex uses the length of the well screen intersecting each model layer to determine the proportion of the total well pumping rate assigned to each well grid cell in the model, even though MODFLOW considers a cell to be screened over its entire vertical length, regardless of the length of screen assigned to the cell. The following equation is used to calculate the pumping rate for each grid cell.
\[ Q = \frac{L_i K_x}{\sum (L_i K_x)_i} Q_T \]

where:
- \( Q_i \) is the discharge from layer \( i \) to a particular well in a given stress period,
- \( Q_T \) is the well discharge in that stress period,
- \( L_i \) is the screen length in layer \( i \),
- \( K_x \) is the hydraulic conductivity in the x-direction in layer \( i \), and
- \( \sum (L_i K_x)_i \) represents the sum of the products of screen length and hydraulic conductivities in the x-direction of all layers penetrated by the well.

This approach, in which a multi-layer well is represented as a group of single layer wells, fails to take into account the inter-connection between various layers provided by the well. One of the most significant problems related to this approach is that well grid cells are essentially shut off when the water table drops below the bottom of the grid cell (i.e. when the grid cell becomes dry). This automatically reduces the total pumping rate of the well, and may cause the water table to “rebound” and re-activate the well grid cell. This type of on-again-off-again behavior for the pumping well(s) causes the solution to oscillate, and may prevent the model from converging to a solution. In the event the model does converge to a solution, the model results may be misleading if one or more pumping wells have lower than expected total pumping rates.

Alternately, a multi-layer well can sometimes be simulated by means of a vertical column of high permeability cells with a screen at the bottom of the column. In this case, if the top part of the model becomes dry, the total pumping rate is unaffected. This also takes into account the vertical inter-connection between layers. The downside of this approach is that the conductivity contrasts can lead to convergence problems.

Both the MODFLOW MNW package and MODFLOW-SURFACT FWL package provide better representation of wells screened over multiple layers. Both of these packages are able to dynamically redistribute pumping rates to the remaining active grid cells if one or more cells in the screened interval goes dry, thereby more accurately simulating the real-world effects of partial overpumping of a well screened over multiple layers.

**Multiple Wells in the Same Cell**

Visual MODFLOW Flex supports multiple pumping wells in the same cell, a common scenario when grid cell sizes are large and wells are close to one another. When the .WEL package is created, and there are multiple wells in the same cell, you will see multiple entries for that cell; the well name will appear as a last parameter, and this will allow you to confirm that the correct wells have made it into the correct cells; an example is below:

in Layer 2, Row 40, Col 40, if this cell has 2 wells in model (named PW1 and PW2), in the WEL package this will appear as:

```
2  40  50  -100  #PW1
2  40  50  -250  #PW2
```
7.6.4 Define Wall (HFB)

Visual MODFLOW Flex supports the Horizontal Flow Barrier (HFB) Package included with MODFLOW. The Horizontal-Flow-Barrier (HFB) Package, or Wall Boundary as it is referred to in Visual MODFLOW Flex, was developed to simulate thin, vertical, low-permeability features that impede the horizontal flow of groundwater. This package allows such features to be simulated without the need to reduce grid spacing in an excessive number of model cells, thus enhancing model efficiency. These features are approximated as a series of horizontal flow barriers conceptually situated on the boundaries between adjacent cells in the finite difference grid.

The key assumption underlying the HFB Package is that the thickness of a barrier is negligibly small in comparison to the horizontal dimensions of the cells in the grid. Barrier width is not explicitly considered in the Package, but is represented in MODFLOW as a hydraulic characteristic defined as barrier hydraulic conductivity divided by barrier thickness.

The HFB Package requires the following information as input for each cell containing this boundary condition:

- \((I_1, J_1)\): Row and column number of grid cell on one side of the barrier.
- \((I_2, J_2)\): Row and column number of grid cell on the other side of the barrier.
- Hydchr: Hydraulic characteristic of the barrier (equivalent to hydraulic conductivity of the barrier divided by the width of the barrier).

Wall Boundaries are currently not supported for MODFLOW-USG or MODFLOW-LGR numerical models, or MODFLOW-2000/2005 grids using a Uniform or Semi-Uniform vertical layering approach.

Walls can be defined in the Conceptual Model workflow, or in existing numerical models using structured grids (excluding MODFLOW-LGR). The behaviour for Walls is similar to other boundary conditions in Visual MODFLOW Flex; if you create this object in the Conceptual Model workflow, then the numerical realization of the Wall will be created after you run "Convert to Numerical Model" steps. If you "Assign" a wall in a numerical model, then Visual MODFLOW Flex will create the conceptual object and also create the numerical wall representation at the same time.

In Visual MODFLOW Flex, Wall objects are assembled using existing data objects in the Data tree and from Conceptual Model. Horizontal Flow Barriers (or Walls) are assigned by using polyline objects to define the spatial extent, and Horizons to define the upper and lower extents.
How to Create a Wall Boundary

1. Select "Define Wall Boundary" from the conceptual model, or select "Assign" option at the Define Boundary conditions step in the numerical model.
2. Import the desired polyline object, or digitize the desired polyline using the 2D Viewer (see Creating New Data Objects for more details). Note: the polyline data object must contain only one line feature.
3. Choose this Wall object from the Data tree.
4. Click on the button (under Select Polyline) in the Define Wall Boundary Condition window to apply this data.
5. In the Model Explorer, expand the Model Explorer tree, then expand Structure / Horizons to see a list of Horizons.
6. Select a Horizon object from the Model explorer that represents the top location for the wall.
7. Click on the button (under Top Horizon) in the Define Wall Boundary Condition window to apply this data.
8. Repeat these steps for the bottom Horizon.
9. Define the Attributes:
   - Conductivity: The hydraulic conductivity of the barrier wall.
   - Thickness: The thickness of the barrier wall.
10. Click OK when you are finished. The Wall object will appear on the Model explorer, under the Conceptual Model / Boundary Conditions folder. If you created the Wall from within a numerical model workflow, then you will also see an item "Wall". If you created the Wall in the Conceptual Model, then the Wall numerical boundary condition will only show up after you complete the Conceptual to Numerical Model Conversion.

Visual MODFLOW Flex will automatically calculate the appropriate cell faces based on the intersection point of the polyline and spanning through the appropriate model layers.

**Conceptual Model Wall Object**  
**Numerical Model Wall Object**

---

**Edit Conceptual Model Wall Object**

You can edit the attributes for a Conceptual Model Wall object by right-clicking on this item in the Model Explorer, and selecting "Edit Boundary Condition". Please note that if you change the attributes, you need to re-run the Conceptual Model to Numerical Model conversion in order to re-generate the numerical model Wall object(s).

**Edit Numerical Model Wall Attributes**

You can edit the attributes for a Numerical Model Wall object by right-clicking on this item in the Model Explorer, and selecting "Edit Attributes...". The attributes will load as a CSV file in either Excel or Notepad (or other text editors if you have associated a different program to .CSV file extensions). Make the desired changes to the values, then close (and Save) the file. When you are finished, and wish to update the Wall object with your edits, right-clicking on this item in the Model Explorer, and selecting "Reload Attributes..."

---

### 7.6.5 Theory

Every model requires an appropriate set of boundary conditions to represent the system’s relationship with the surrounding systems. In the case of a groundwater flow model, boundary conditions will describe the exchange of flow between the model and the external system. In the case of a mass transport model, the boundary conditions will also describe the exchange of solute mass between the model and the external system.
The following sections present an overview of the boundary condition packages supported in Visual MODFLOW Flex. Each section includes a brief description of the boundary condition, including the input data required by MODFLOW and the supported data objects for defining the boundary condition geometry. The following boundary conditions are discussed in this section:

- **Well**
- **Constant Head**
- **River**
- **General Head**
- **Drain**
- **Recharge**
- **Evaporation**
- **Lake**
- **Specified Flux**
- **Time-varying Material Properties**

**Well**

The pumping well boundary condition is used to simulate wells (or other features) that withdraw water from or add water to the model at a constant rate during a stress period, where the rate is independent of both the cell area and head in the cell.

For finite difference translations, Visual MODFLOW Flex uses the Well (WEL) package, provided with MODFLOW. The MODFLOW input data for Well cells is stored in the `projectname.WEL` file. You can define the location for horizontal or deviated wells, which include the well path and the screen location. When you translate your conceptual model to MODFLOW format, the horizontal well screen location is converted to set of pumping well cells side-by-side. Another option is to define a specified flux or drain boundary condition in VMOD Flex. These are the workarounds, since there is no MODFLOW package for horizontal wells.

For finite element model translations, VMOD Flex translates the pumping well boundary conditions as a Type 4 (Well) boundary condition. Please note that the defined screen interval must extend beyond half of the element height for it to be assigned the boundary condition.

Currently, deviated/horizontal well translation is not supported for finite element models. If you intend to translate to FEFLOW, please make sure all pumping well boundary conditions are defined using vertical wells.

**Required Data**

In VMOD Flex, pumping well boundary conditions are defined using the well data contained in a wells data object. During the boundary condition creation process, you will be required to select a wells data object from the Data Explorer.

A well can only be used if it meets the following requirements:

- The pumping well must be located within the simulation domain
- A screen must be defined for the pumping well
- A pumping schedule must be defined for the pumping well
For information on importing well data, please see "Importing Wells" section. For information on defining well data for existing wells data objects, please see the “Well Table” section.

**Constant Head**
Currently, this boundary condition is only supported for Finite Difference Model translation.

The Specified Head boundary condition, also known as Constant Head in VMOD Flex, is used to fix the head value in selected grid cells regardless of the system conditions in the surrounding grid cells, thus acting as an infinite source of water entering the system, or as an infinite sink for water leaving the system. Therefore, *specified head boundary conditions can have a significant influence on the results of a simulation, and may lead to unrealistic predictions, particularly when used in locations close to the area of interest.*

During translation, VMOD Flex uses the Time-Variant Specified-Head Package provided with MODFLOW. The MODFLOW input data for Specified Head cells is stored in projectname.CHD file.

Unlike most other transient MODFLOW boundary condition packages, the Specified-Head package allows the specified heads to be linearly interpolated in time between the beginning and end of each stress period, such that the specified head for a grid cell may change at each time step of a given stress period. If the simulation is steady-state, the specified starting head value will be used.

**Required Data**
The Specified-Head package requires the following information for each specified head grid cell for each stress period:

- **Start Head**: Specified head value at the beginning of the stress period
- **Stop Head**: Specified head value at the end of the stress period

**Supported Geometry**
The geometry for Specified Head boundary conditions can be specified using Polylines or Polygons.
**A Note on Leakance vs. Conductance**

Some Type-3 (Head-Dependent Flux) boundary conditions (e.g. river and general head) require defining a conductance parameter (for MODFLOW). Conductance is a numerical parameter representing the resistance to flow between the cell assigned with that boundary condition and the surrounding cells. Conductance between cells is calculated using some average hydraulic conductivity of the cells, the area of the interface between the cells and the distance between the cell centers. The Conductance calculation requires the cell geometry (cell interfaces).

In Visual MODFLOW Flex, when you create a new conceptual boundary condition, this is done using shapes: polyline, polygon, and side faces. At this point, there is no notion of cell geometry, as a result, the conductance cannot be calculated since the cell face area cannot yet be calculated. For this reason, you are asked to define "Leakance" instead of "Conductance". Leakance is a conceptual (hydrogeological) term, and is expressed per unit area (if the conceptual boundary condition object is a polygon) or per unit length (if the conceptual boundary condition object is a polyline).

When you create the boundary condition, the Leakance can be calculated based on other defined parameters, or it can be explicitly defined.

When you look at the numerical representation (cell realization) of the boundary condition, you will see "Conductance" as the parameter, since this value can be calculated based on the intersecting cell geometry. For a boundary condition assigned with a polygon or side face, the Leakance is multiplied by the cell area, in order to get Conductance. For a boundary condition assigned with a polyline, the Leakance is multiplied by the length of the line that intersects the cell, in order to get the Conductance.


**River**

The River boundary condition is used to simulate the influence of a surface water body on the groundwater flow. Surface water bodies such as rivers, streams, lakes and swamps may either contribute water to the groundwater system, or act as groundwater discharge zones,
depending on the hydraulic gradient between the surface water body and the groundwater system.

For finite difference models, VMOD Flex uses the River Package included with MODFLOW. The MODFLOW input data for River grid cells is stored in `projectname.RIV` file. Currently, translation of river boundary conditions is not supported for finite element (FEFLOW) translations.

The MODFLOW River Package simulates the surface water/groundwater interaction via a seepage layer separating the surface water body from the groundwater system (see following figure).

![Schematic Diagram](image)

**Required Data**
The MODFLOW River Package input file requires the following information for each grid cell containing a River boundary:

- **River Stage**: The free water surface elevation of the surface water body. This elevation may change with time.
- **Riverbed Bottom**: The elevation of the bottom of the seepage layer (bedding material) of the surface water body.
- **Conductance**: A numerical parameter representing the resistance to flow between the surface water body and the groundwater caused by the seepage layer (riverbed).

The Conductance value (C) may be calculated from the length of a reach (L) through a cell, the width of the river (W) in the cell, the thickness of the riverbed (M), and the vertical hydraulic conductivity of the riverbed material (K) using the following formula:
For situations where the River package is used to simulate lakes or wetlands, the L and W variables would correspond to the X-Y dimension of the River boundary grid cells.

When a River boundary condition is assigned, the Use default Leakance option is automatically selected.

If the Use default Leakance option is selected, the River boundary condition requires the following data:

- **River Stage**: The free water surface elevation of the surface water body. [L]
- **Riverbed Bottom**: The elevation of the bottom of the seepage layer (bedding material) of the surface water body. [L]
- **Riverbed Thickness**: Thickness of the riverbed (seepage layer) [L].
- **Leakance**: A numerical parameter representing the resistance to flow between the surface water body and the aquifer (this field is read-only and is calculated using the formula described below). [L^2/T]
- **Riverbed Kz**: Vertical hydraulic conductivity of the riverbed material. [L/T]
- **River Width**: Width of the river. [L]

When a polyline is used to define the river geometry, the default conductance formula is as follows:

\[
C = \frac{K \times L \times W}{M}
\]

When a polygon is used to define the river geometry, the default conductance formula is as follows:

\[
SCOND = \frac{SRCHLNG \times WIDTH \times K \times SUCOCOND}{RBTHICK}
\]

Where:

- `$COND`: is the Conductance
- `$SRCHLNG`: is the reach length of the river line in each grid cell
- `$WIDTH`: is the River Width in each grid cell
- `$K`: is the Riverbed Kz
- `$RBTHICK`: is the thickness of the riverbed
- $\text{SUCTOCOND}$: is the conversion factor for converting the $K$ value to the same L and T units used by $\text{COND}$
- $\text{SRBHICK}$: is the Riverbed Thickness
- $\text{DX}$: is the length of each grid cell in the X-direction
- $\text{DY}$: is the length of each grid cell in the Y-direction

If the Use default Leakage option is turned off, the fields used for calculating the River Leakage value (Riverbed Thickness, Riverbed Kz, and River Width) are removed from the table, and the Leakage field becomes a writable field where a value may be entered.

**Supported Geometry**
The geometry for River boundary conditions can be specified using polylines or polygons

**General Head**
For finite difference models, VMOD Flex supports translation of the General-Head Boundary Package included with MODFLOW. The MODFLOW input data for General-Head grid cells is stored in the projectname.GHB file. Currently, for finite element models, translation of this boundary condition is not supported.

The function of the General-Head Boundary (GHB) Package is mathematically similar to that of the River, Drain, and Evapotranspiration Packages. Flow into or out of a cell from an external source is provided in proportion to the difference between the head in the cell and the reference head assigned to the external source. The application of this boundary condition is intended to be general, as indicated by its name, but the typical application of this boundary condition is to represent heads in a model that are influenced by a large surface water body outside the model domain with a known water elevation. The purpose of using this boundary condition is to avoid unnecessarily extending the model domain outward to meet the element influencing the head in the model. As a result, the General Head boundary condition is usually assigned along the outside edges (sides) of the simulation model domain. This scenario is illustrated in the following figure.
The primary differences between the General-Head boundary and the Specified Head boundary are:

- the model solves for the head values in the General-Head grid cells whereas the head values are specified in Constant Head cells.
- the General-Head grid cells do not act as infinite sources of water whereas Specified Head cells can provide an infinite amount of water as required to maintain the specified head. Therefore, under some circumstances, the General-Head grid cells may become dry cells.

**Required Data**
The General-Head Boundary Package requires the following information for each General-Head grid cell:

- Stage: This is the head of the external source/sink. This head may be physically based, such as a large lake, or may be obtained through model calibration.
- Conductance: The is a numerical parameter that represents the resistance to flow between the boundary head and the model domain.

In contrast to the River, Drain, and Evapotranspiration packages, the General Head package provides no limiting value of head to bind the linear function in either direction. Therefore, as the head difference between a model cell and the boundary head increases/decreases, flow into or out of the cell continues to increase without limit. Accordingly, care must be used to ensure that unrealistic flows into or out of the system do not develop during the simulation.
The leakance value may be physically based, representing the conductance associated with an aquifer between the model area and a large lake, or may be obtained through model calibration. The leakance value ($C$) for the scenarios illustrated in the preceding figure may be calculated using the following formula:

$$\frac{(L \times W) \times K}{D}$$

where

- $(L \times W)$ is the surface area of the grid cell face exchanging flow with the external source/sink
- $K$ is the average hydraulic conductivity of the aquifer material separating the external source/sink from the model grid
- $D$ is the distance from the external source/sink to the model grid

When a General-Head boundary condition is assigned, the "Use default leakance" option is automatically selected.

If the "Use default leakance" option is selected, the General-Head boundary condition requires the following data:

- **Stage**: The head value for the external source/sink
- **Leakance**: A numerical parameter representing the resistance to flow between the boundary head and the model domain (this field is read-only and is calculated using formula described below)
- **Distance to Reservoir**: The distance from the external source/sink to the General-Head grid cell
- **General Head Average Conductivity**: The average hydraulic conductivity of the aquifer material separating the external source/sink from the model grid

The default formula used to calculate the Leakance value for the General-Head boundary is:

$$\text{COND} = \frac{\text{KAVG} \times \text{FACEAREA} \times \text{UCTOCOND}}{\text{DIST}}$$

where

- $\text{COND}$: is the Leakance for each General-Head grid cell
- $\text{KAVG}$: is the Average Conductivity
- $\text{FACEAREA}$: is the surface area of the selected grid cell Face for each General-Head grid cell (automatically calculated during translation)
- $\text{UCTOCOND}$: is the conversion factor for converting the $K$ value to the same Length (L) and Time (T) units used by $\text{COND}$
**$DIST:** is the Boundary Distance, the distance from the external source to the assigned general head boundary

If the "Use default conductance" formula option is not selected, the fields used for calculating the General-Head Conductance value (Distance to Reservoir, Average Conductivity) are removed from the table, and the Leakance field becomes a writable field where a value may be entered.

**Supported Geometry**
The geometry for General-Head boundary conditions can be specified using a polygon data objects.

**Drain**
For finite difference models, VMOD Flex supports the standard Drain Boundary Package included with MODFLOW. The MODFLOW input data for Drain grid cells is stored in the projectname.DRN file. Currently, for finite element model translation, this boundary condition is not supported.

MODFLOW's Drain Package is designed to simulate the effects of features such as agricultural drains, which remove water from the aquifer at a rate proportional to the difference between the head in the aquifer and some fixed head or elevation. The Drain package assumes the drain has no effect if the head in the aquifer falls below the fixed head of the drain.

**Required Data**
The Drain Package requires the following information as input for each cell containing this boundary condition:

- **Elevation:** The drain elevation, or drain head of the free surface of water within the drain. The drain is assumed to run only partially full, so that the head within the drain is approximately equal to the median elevation of the drain.
- **Leakance:** The drain leakance is a lumped coefficient describing the head loss between the drain and the groundwater system. This loss is caused by converging flow patterns near the drain, the presence of foreign material around the drain, channel bed materials, the drain wall, and the degree to which the drain pipe openings may be blocked by chemical precipitates, plant roots, etc.

There is no general formulation for calculating drain leakance. In most situations, the detailed information required to calculate drain leakance is not available to the groundwater modeler. These details include the detailed head distribution around the drain, aquifer hydraulic conductivity near the drain, distribution of fill material, number and size of the drain pipe openings, the amount of clogging materials, and the hydraulic conductivity of clogging materials. It is common to calculate drain leakance from measured values of flow rate and head difference. Drain leakance value is usually adjusted during model calibration.

When a polyline is used to define the boundary condition geometry, the default formula for the leakance is as follows:
When a polygon is used to define the boundary condition geometry, the default leakance formula is as follows:

\[ $COND = RCHLNG \times LCOND $\]

where

- $COND$: is the Leakance
- $RCHLNG$: is the reach length of the drain in each grid cell
- $LCOND$: is the Leakance per unit length of the drain in each grid cell
- $SCOND$: is the Leakance per unit area of the drain in each grid cell
- $DX$: is the length of each grid cell in the X-direction
- $DY$: is the length of each grid cell in the Y-direction

If the Use default leakance option is turned off, the fields used for calculating the Drain Leakance value (Leakance per unit length or area) are removed from the table and the Leakance field becomes a read/write field where any value may be entered.

**Supported Geometry**

The geometry for General-Head boundary conditions can be specified using polygon or polylines.

**Recharge**

For finite difference models, Visual MODFLOW Flex supports the Recharge Package (RCH) included with MODFLOW. The Recharge input data for MODFLOW is stored in the projectname.RCH file. For finite element models, recharge boundary conditions are translated as the In(+) / Out(-) flow material parameter.

The recharge boundary condition is typically used to simulate surficially distributed recharge to the groundwater system. Most commonly, recharge occurs as a result of precipitation percolating into the groundwater system. However, the recharge boundary can potentially be used to simulate recharge from sources other than precipitation, such as irrigation, artificial recharge, or seepage from a pond.

**Please Note:** The recharge rate is a parameter that is not often measured at a site, but rather, it is assumed to be a percentage of the precipitation. This percentage typically ranges from 5% to 20% depending on many different factors including:

- the predominant land use and vegetation type,
- the surface topography (slope), and
- the soil cover material
Required Data
The Recharge Package requires the following information as input for each cell containing this boundary condition:

- **Recharge (L/T)**: The input flux due to recharge. The recharge is applied in one of three ways as part of the package translation settings: 1. recharge is applied only to cells in layer 1, 2. recharge is applied to cells the specified layer, 3. recharge is applied to the upper-most active cell. See the documentation in the Recharge Translation Settings documentation for more details.

- **Ponding (L)** [optional, only used in the MODFLOW-SURFACT engine]: The maximum allowable depth of the water table above the ground surface. Calculated heads elevations above this depth (+ground surface) are truncated by the model.

ℹ️ **Please Note**: MODFLOW-SURFACT simply removes excess ponded water from the model without routing it to other cells.

**Supported Geometry**
The geometry for Recharge boundary conditions can be specified using polygon data objects and by points, and for an entire layer.

**Evapotranspiration**
For finite difference models, Visual MODFLOW Flex supports the Evapotranspiration Package (ET) included with MODFLOW. After translation, the Evapotranspiration input data for MODFLOW is stored in the projectname.EVT file. Currently, this boundary condition is not supported for finite element translation.

The evapotranspiration boundary condition simulates the effects of plant transpiration, direct evaporation, and seepage at the ground surface by removing water from the saturated groundwater regime.

The evapotranspiration boundary approach is based on the following assumptions:

- When the water table is at or above the ground surface (top of layer 1), evapotranspiration loss from the water table occurs at the maximum rate specified by the user.
- When the elevation of the water table is below the ‘extinction depth’, or is beneath layer 1, evapotranspiration from the water table is negligible.

Between these limits, evapotranspiration from the water table varies linearly with water table elevation.

**Required Data**
The Evapotranspiration Package requires the following information:

- **Evapotranspiration rate**: The rate of evapotranspiration as it occurs when the water table elevation is equal to the top of the grid cell elevation. This value should be entered in the units set for recharge as defined in the Project Settings.
• Extinction Depth: The depth below the top of grid cell elevation where the evapotranspiration rate is negligible.

The Evapotranspiration Package approach is based on the following assumptions:

When the water table is at or above the ground surface (top of layer 1), evapotranspiration loss from the water table occurs at the maximum rate specified by the user.
When the elevation of the water table is below the ‘extinction depth’, or is beneath layer 1, evapotranspiration from the water table is negligible.
Between these limits, evapotranspiration from the water table varies linearly with water table elevation.

Supported Geometry
The geometry for Evapotranspiration boundary conditions can be specified using polygon data objects

Lake
For finite difference models, Visual MODFLOW Flex supports the Lake (LAK3) package for MODFLOW. After translation, the Lake input data for MODFLOW is stored in the projectname.LAK file. Currently, translation of this boundary condition is not supported for finite element models.

The lake boundary condition can be used to simulate the effects of stationary surface-water bodies such as lakes and reservoirs on an aquifer. The lake boundary is an alternative to the traditional approach of using the general head boundary condition. The main difference in the lake boundary is that the lake stage is calculated automatically based on the water budget, which is a function of inflow, outflow, recharge, etc.

For more information on the Lake package, please refer to USGS publication, Documentation of a Computer Program to Simulate Lake-Aquifer Interaction Using the MODFLOW Ground-Water Flow Model and the MOC3D Solute-Transport Model.

Required Data
The lake package requires the following input parameters:

• Stage [L]: The initial stage of the lake at the beginning of the run.
• Bottom [L]: The elevation of the bottom of the seepage layer (bedding material) of the surface water body.
• Leakance \( [L^2/T] \): A numerical parameter representing the resistance to flow between the boundary head and the model domain (this field is read-only and is calculated using formula described below)
• Lakebed Thickness [L]: Thickness of the lakebed (seepage layer).
• Lakebed Conductivity [L]: Vertical hydraulic conductivity of the lakebed material.
• Precipitation Rate per Unit Area \([L/T]\): The rate of precipitation per unit area at the surface of the lake.
• Evaporation Rate per Unit Area \([L/T]\): The rate of evaporation per unit area from the surface of the lake.
- **Overland Runoff** \([L^3/T]\): Overland runoff rate from an adjacent watershed entering the lake.
- **Artificial Withdrawal** \([L^3/T]\): The volumetric rate of water removal from a lake by means other than rainfall, evaporation, surface outflow, or ground-water seepage. Normally, this would be used to specify the rate of artificial withdrawal from a lake for human water use, or if negative, artificial augmentation of a lake volume for esthetic or recreational purposes.

The default leakance formula is as follows:

\[
$COND = \frac{SDX \times SDY \times K \times UCTOCOND}{RBTHICK}
\]

where

- $COND$: is the Leakance
- $K$: is the Lakebed Kz
- $UCTOCOND$: is the conversion factor for converting the $K$ value to the same L and T units used by $COND$
- $RBTHICK$: is the Lakebed Thickness
- $DX$: is the length of each grid cell in the X-direction
- $DY$: is the length of each grid cell in the Y-direction

If the Use default Leakance option is turned off, the fields used for calculating the River Conductance value (Lakebed Thickness, Lakebed Kz) are removed from the table, and the Leakance field becomes a writable field where a value may be entered.

**Supported Geometry**
The geometry for Lake boundary conditions can be specified using polygon data objects.

**Specified Flux**
When you create the flux object, the value you define should be the flux per unit length (when using polyline) or per unit area (for polygon). When the cell realization is calculated, then VMOD Flex will calculate the flux for each cell based on the length of the line that passes thru the cell (for polyline) or the cell area (for polygon). For these reasons, you will see a significant difference in the parameter values for the conceptual object (shape) vs. the summed values for all cells (the cell realization).

For finite difference models, VMOD Flex supports the Specified Flux (FHB1) package for MODFLOW. After translation, the specified flux input data for MODFLOW is stored in the projectname.FHB file. Currently, translation of this boundary condition is not supported for finite element models.

The Specified Flux boundary condition allows you to specify flow, as a function of time, at selected model cells. FHB1 is an alternative and (or) supplement to the recharge (RCH)
package for simulating specified-flow boundary conditions. The main differences between the FHB1 package and the recharge package are as follows:

- FHB1 package can simulate specified-flux on the top, side, bottom or intermediate layers in the simulation domain, whereas the recharge package can only be applied to the top and intermediate layers.
- FHB1 package allows you to specify a starting flux and an ending flux (for each stress period, if transient). The package then uses linear interpolation to compute values of flow at each model time step.


**Required Data**
The specified flux package requires the following input parameters:

- Starting Flux (L³/T)
- Ending Flux (L³/T)

**Supported Geometry**
The geometry for Specified Flux boundary conditions can be specified using polygon or polyline data objects

**Time-Varying Material Properties**
Time-varying material properties can be specified for transient groundwater flow simulations (using the TMP1 package add-on in MODFLOW-SURFACT). The list of time-dependent properties includes:

- horizontal hydraulic conductivity,
- either vertical hydraulic conductivity or leakance,
- specific storage, and
- specific yield.

The reference values of these properties are assigned in the standard format associated with the BCF4 package. These reference values are then scaled by factors assigned in the TMP1 package, where the factors are time-dependent over the simulation period.

**Required Data**
The time-varying material package requires the following dimensionless input parameters for each stress period specified in the boundary condition:

- **Kxx/Kyy Scaling [-]**: the scaling factor by which the base horizontal conductivities (Kxx and Kyy) values will be multiplied for the associated stress period
- **Kzz Scaling** [-]: the scaling factor by which the base vertical conductivity (Kzz) value will be multiplied for the associated stress period
- **Leakance Scaling** [-]: the scaling factor by which the base vertical leakance values will be multiplied for the associated stress period.
- **Sy Scaling** [-]: the scaling factor by which the base specific yield values will be multiplied for the associated stress period.
- **Ss Scaling** [-]: the scaling factor by which the base specific storage values will be multiplied for the associated stress period.

ℹ️ **Please Note**: the Kzz and Leakance Scaling factors are mutually exclusive and which one is used is based on the settings in the BCF4 package.

**Supported Geometry**
The geometry for time-varying material boundary conditions can be specified using polygon or polyline data objects or on a cell-by-cell basis.
8 Defining Grids/Meshes

At this step, you can create finite difference grids for MODFLOW, unstructured grids for MODFLOW-USG, or finite element meshes for a FEFLOW run.

Quick Overview

Instructions:
- Define Finite Difference Grid;
- Define UnStructured Grid;
- Define Finite Element Mesh

Pre-requisites: The Conceptual Model has been defined.

Result: A numerical grid or mesh is created.

Next Steps:
- Convert the conceptual model to any one of the following:
  - Convert to a MODFLOW (finite difference) model;
  - Convert to a MODFLOW-USG model
  - Convert to a FEFLOW finite element model

Once you have created your conceptual model you can discretize your model using either the finite difference method, unstructured finite volume (Control Volume Finite Difference, CVFD) or the finite element method. The finite difference method involves fitting your conceptual model to one or more finite difference grids. The CVFD method (used in MODFLOW-USG), allows you to conform the grid to your wells and boundary conditions. For MODFLOW-based numerical models, after conversion, the resulting numerical model can be viewed/edited and then simulated in the Visual MODFLOW Flex environment. The finite element method
involves fitting your conceptual model to a finite element mesh. Once translated, a FEFLOW ASCII .FEM file is created (and saved on the hard disk), which can then be opened and simulated using FEFLOW.

Please Note: a license of FEFLOW v.5.4 or above is required to open FEFLOW .FEM files generated from Visual MODFLOW Flex.

Sections covered in this chapter:

- Define Finite Difference Grid
- Define UnStructured Voronoi Grid (V-Grid)
- Define Unstructured QuadTree Grid (Q-Grid)
- Define Fine Element Mesh

8.1 Define Finite Difference Grid

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Specify the horizontal grid resolution and the vertical layering type and resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites:</td>
<td>The Conceptual Model has been defined.</td>
</tr>
<tr>
<td>Result:</td>
<td>A numerical grid is created</td>
</tr>
<tr>
<td>Next Steps:</td>
<td>Convert the conceptual model to a MODFLOW model</td>
</tr>
</tbody>
</table>

From the "Select Grid Type" step in the Conceptual Model workflow, click on the "Define Finite Difference Grid button":

© 2019 by Waterloo Hydrogeologic
Defining the Horizontal Grid

By default, Visual MODFLOW Flex discretizes the horizontal grid using 20 rows and 20 columns, with no rotation. However, you can customize the grid to your liking, by modifying the settings in the horizontal grid dialog (shown below).
Enter a unique Name for the numerical grid. This name will appear in the Model Explorer tree once the grid is created.

The grid can be rotated counter-clockwise about the grid origin by entering a value between 0 and 360 in the Rotation text field.

The Xmin and Ymin values refer to the X-Y coordinates of the bottom-left corner of the numerical grid. The Xmax and Ymax values refer to the X-Y coordinates of the top-right corner of the numerical grid.

The Columns and Rows fields allow you to define the Grid Size.

You can also specify the Cell Height and Cell width and the Rows and Columns (and potentially the extents) will be updated accordingly.

Click the [Next] button to proceed to define the vertical discretization.

**Defining the Vertical Grid**

The first step in defining the vertical grid is selecting the Grid Type. There are three different grid types: Deformed, Uniform and Deformed-Uniform. Each grid type is described in the following sections.

**Grid Types**
Deformed

In a deformed grid, the tops and bottoms of the model layers conform to the horizons elevation. You can refine the model layers, by diving the structural zones into proportionately thick layers.

Cross sectional view of deformed grid from Visual MODFLOW Flex

A Minimum Cell Thickness must be specified as MODFLOW does not permit lateral discontinuity of layers, i.e., a layer cannot have a thickness of 0 at any point in the layer. When horizons are on-lapping one another, resulting in a zero cell thickness, the minimum cell thickness is applied and the horizons are shifted based on the horizon types defined in the Horizon settings (See “Horizon Types” section).

For deformed grids, you have the option of refining (subdividing) each layer into a specified number of equally thick layers. In the table located below the grid description, enter a refinement factor for the desired layer(s). For example, a layer refinement factor of 2 would subdivide the layer into two equally spaced layers.

<table>
<thead>
<tr>
<th>Layer Name</th>
<th>Layer Refinement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zone1</td>
<td>2</td>
</tr>
<tr>
<td>Zone2</td>
<td>2</td>
</tr>
<tr>
<td>Zone3</td>
<td>1</td>
</tr>
</tbody>
</table>

After entering a refinement factor, click the [Apply] button to view the changes in the adjacent 2D Viewer.
**Uniform**

In a uniform grid, a number of layers with uniform thickness will be created. At the time of translating the conceptual model to the numerical model, the properties will be assigned to the appropriate grid cells to represent the geological structure. This grid is useful for transport or density-dependent simulations, where it is desirable to have fine vertical discretization.

Cross sectional view of uniform grid from Visual MODFLOW

When this grid type is selected, specify the number of layers to create in the Number of Layers field (default is 10).

ℹ️ **Please Note:** Maximum number of vertical layers is 1000.

**Semi-Uniform**

In a semi-uniform grid, the top and bottom of the grid are deformed, following the top-most and bottom-most horizons respectively; in between, a set of uniformly thick layers will be generated. At the time of translating the conceptual model to the numerical model, the properties will be assigned to the appropriate grid cells to represent the geological structure. This grid is useful where you have discontinuous layers.
For Deformed-Uniform grids, you must specify a Minimum Cell Thickness (see above) and the Number of Layers.

Once the grid is created, it will appear as a new node in the tree as shown below:
8.1.1 Edit Grid

When a numerical grid is initially created (see "Creating a Finite Difference Grid" section), the horizontal grid spacing is uniform. However, in many situations it is advantageous to have non-uniform grid spacing to allow for finer grid discretization in the areas of interest, and larger grid spacing in areas which are less important, or where less data is available. Visual MODFLOW Flex allows you to refine or coarsen areas of a numerical grid by adding or removing grid lines within a specified row/column interval. This process is described in the following section.

Note

You cannot edit a grid that contains a child grid (i.e. grid edits are not currently available for LGR Models).

To edit the grid (rows and columns) select the View/Edit Grid step of the Numerical Workflow and then select the Edit grid button. Alternatively, you can find this option from the Model Explorer tree, right-click on the Numerical Grid, and select Edit Numerical Grid from the pop-up menu.

The Edit grid dialog provides options for refining/coarsening a numerical grid.

When you edit the grid by default you are creating a new grid - and a default New grid name is generated (you can adjust this if you wish). If however, you prefer to simply overwrite your existing grid with the changes you make simply select the check box to do so.
Tip! You can add data objects (e.g., boundary conditions, wells, site maps) from the data explorer to the 2D Viewer preview to assist you in determining the areas in which horizontal refinement/coarsening should be applied. Select the desired data objects from the Data Explorer, and the data will be displayed in the 2D Viewer preview.

- Select the Edit Rows radio button to add/remove grid lines (rows) along the Y-direction of the numerical grid, or select the Edit columns radio button to add/remove grid lines (columns) along the X-direction of the numerical grid.

- Specify the row/column interval for which the refinement/coarsening should be applied, by specifying the From row/column and the To row/column. For example, if you would like to refine the grid area between row 20 and 30, you would enter 20 as the From row and 30 as the To row. In the preview the blue line represents the From row/column while the yellow line represents the To row/column.
Next, specify the refinement/coarsening factor. In the example above we are going to Replace every 1 row with 2 - so the grid will be refined. This means that we would subdivide each row within the specified interval into two equally spaced rows.

Instead, if we changed the option to indicate we are going to Replace every 2 rows with 1 then the grid would be coarsened. This means we would reduce the number of rows within the specified interval by a factor of 2.

When you select the Apply grid edit button the preview will be updated with your edits. You also have options to Undo your edit - this will undo your last change. You can keep selecting Undo edit to undo more changes. Reset will return the preview to the original rows and columns.

Click the OK button to create the new refined/coarsened grid. You will find it in the Model Explorer.

When editing a grid all inputs from the original grid (property zones, boundary conditions, pumping wells, observation wells, zone budget zones, and particles) will be reassigned as appropriate to the new edited grid. If the original grid has multiple runs associated with it - the inputs are taken from the first run.

Quality Indicators
The quality indicators can help you find areas in your grid where you may wish to make adjustments. By default the Maximum ratio threshold is 1.5 however you can adjust this to your model requirements. If you do not wish to have the Quality indicators on simply uncheck the Highlight cells exceeding threshold option.

Also displayed in the Minimum and Maximum cell size.

![Quality indicators](image)

Script
If preferred, you can make your grid edits by using the Script option. On this tab you will find the script for any edits you make on the Grid edit details tab allowing you to learn the script language.
8.1.2 Edit Layers

To edit the grid (layers) select the View/Edit Grid step of the Numerical Workflow and then select the Edit layers button. Alternatively, you can find this option from the Model Explorer tree, right-click on the Numerical Grid, and select Edit Grid Layers from the pop-up menu.

The Edit layers dialog provides options for refining/coarsening/editing a numerical grid. When you edit the grid by default you are creating a new grid - and a default New grid name is generated (you can adjust this if you wish at the time the new grid is generated - once a grid is created, it cannot be renamed). If however, you prefer to simply overwrite your existing grid with the changes you make, simply select the check box to do so.

The dialog also provides you with a preview of the grid layers as viewed by a Row or a Column - you can select this from the Viewing drop down options above the preview window. You can then adjust which Row or Column you are viewing using the up and down arrows as well as the Exaggeration.

When you select the Apply edit button the preview will be updated with your edits. You also have options to Undo your edit - this will undo your last change. You can keep selecting Undo edit to undo more changes. Reset will return the preview to the original rows and columns.
You can add data objects (e.g., boundary conditions, wells, etc.) from the data explorer to the preview to assist you in determining the areas in which refinement/coarsening should be applied. Select the desired data objects from the Data Explorer, and the data will be displayed in the preview.

Several options are available for editing the layers of your grid including:

- **Refine**
- **Insert**
- **Remove**
- **Replace**, and via
- **Script**

These options are described below.

**Refine**

You can select the layer from and to values and then provide a Factor to refine by. For example if you have 8 layers and indicate the From 1 To 8 and a factor of 2 (as shown below) when you select the Apply edit button you would end up with 16 layers (as each of the original 8 layers would be split into 2).
Insert

You can select to insert a new layer using the following options:

- Fraction of layer
- Thickness below layer ceiling
- Thickness above layer floor
- At elevation

When you select which option you want to use you will see in the preview that the selected layer is highlighted in green and when you hover your mouse over the preview the proposed new layer to be inserted will appear as a red line. When you adjust the layer - the preview is adjusted.

On this tab you also have the following Push up and Push down options.

- When neither Push up nor Push down are selected then the bottom surface pushes up the new surface and the top surface pushes down the surface to insert in order for the values of the surface to obey the Minimum layer thickness criteria.

- When Push up is selected then the new surface will Push up all the surfaces (or surface subsets) above this new surface in order for the Minimum layer thickness criteria to be obeyed.
• When Push down is selected then the new surface will Push down all the surfaces (or surface subsets) below this new surface in order for the Minimum layer thickness criteria to be obeyed.

• When both Push up and Push down are selected then the new surface will combine the behavior of when Push up or Push down is selected - it will push all surfaces below and above the new surface enforcing the Minimum layer thickness criteria.

**Remove**
On this tab you can simply select the layer you wish to remove from the preview window or you can specify the Top or Bottom of the layer indicated is to be removed.

**Replace**
This tab provides the option to replace either the top or the bottom of a layer with an existing surface (selected from the Data Explorer) or at a specified elevation.
On this tab you also have the following Push up and Push down options:

- When neither Push up nor Push down are selected then the bottom surface pushes up the new surface and the top surface pushes down the surface to insert in order for the values of the surface to obey the Minimum layer thickness criteria.

- When Push up is selected then the new surface will Push up all the surfaces (or surface subsets) above this new surface in order for the Minimum layer thickness criteria to be obeyed.

- When Push down is selected then the new surface will Push down all the surfaces (or surface subsets) below this new surface in order for the Minimum layer thickness criteria to be obeyed.

- When both Push up and Push down are selected then the new surface will combine the behavior of when Push up or Push down is selected - it will push all surfaces below and above the new surface enforcing the Minimum layer thickness criteria.

**Script**

Similar to the grid edits - if preferred you can make your layer edits by using the Script option. On this tab you will find the script for any edits you make on the Layer edit details tabs allowing you to learn the script language.
8.1.3 Define Child Grid (for LGR)

A higher-resolution block-shaped child grid can be created within any numerical grid (parent grid) and used for running Local Grid Refinement (LGR) simulations with the MODFLOW-2005 LGR package. Refined child grids are often used to improve simulation accuracy around areas of interest within your simulation domain. For example, refined grids may be needed in:

- Regions where hydraulic gradients change substantially over short distances, as would be common near pumping or injecting wells, rivers, drains, and focused recharge.
- Regions of site-scale contamination within a regional aquifer where simulations of plume movement are of interest.
- Regions requiring detailed representation of heterogeneity, as may be required to simulate faults, lithologic displacements caused by faulting, fractures, thin lenses, pinch outs of geologic units, and so on.

Visual MODFLOW Flex allows you to create up to nine child grids within a single parent grid. However, you cannot create a child grid within a child grid, and child grids cannot not overlap and must have at least one parent grid cell in between them.

Adding Child Grids

Child Grids can only be added in the conceptual modeling workflow. Visual MODFLOW Flex currently does not support adding Child Grids onto models that have been imported. If you wish to run LGR on an existing project, you can re-build this using the Conceptual Modeling workflow.

For more information on the local grid refinement package, please refer to the online guide for MODFLOW-LGR and to the published documentation by Mehl and Hill (2005, 2013).

There are two ways in which you can define a child grid within a numerical model grid:

- When defining the parent grid; check the Create Child Grid box in the Vertical Grid dialog, and click the [Next] button.
- After the parent grid has been defined; right-click on the numerical grid from the Model Explorer tree, and select Create Child Grid.

Horizontal Grid Refinement

Horizontal child grid refinement involves specifying the location of the child grid within the parent grid, and defining the row and column refinement ratio.
You can add data objects (e.g., boundaries, wells, site maps) from the data explorer to the 2D Viewer preview to assist you in determining the placement of the child grid within the parent grid. Select the desired data objects from the data explorer, and the data will be displayed in the 2D Viewer preview.

Enter a unique name for the child grid in the Name field.

Next, select the refinement ratio from the Ratio combo box. A ratio of 3:1, for example, will refine the parent cell by a factor of three, resulting in nine horizontal child cells within one parent cell.

Finally, specify the Row Refinement interval and the Column Refinement interval, by selecting the starting row/column and ending row/column, for where the grid refinement should be applied within the parent grid. The child grid can be placed anywhere within the parent grid as long as it does not overlap another child grid.

Please Note: The child grid cannot be rotated; it must be in the same orientation as the parent grid.

Click the [Preview] button to preview the child grid in the adjacent 2D Viewer.
Click the [Next] button to proceed to the next step.

**Vertical Grid Refinement**

Vertical grid refinement involves selecting which model layers to refine and specifying the refinement ratio for the selected layers.

The top of the child grid must always coincide with the parent grid and therefore the Start layer will always be 1. However, the End layer can be any model layer below the top model layer in the simulation domain.

There are two options for defining the refinement ratio. Select Globally for all layers to assign a single refinement ratio to all layers. Alternatively, select Specify each layer to assign a refinement ratio layer by layer.

⚠️ **Please Note**: Although the top layer must be the start layer, vertical refinement does not have to start at the top. Assign a refinement ratio of 1:1 to the top layer and it will not be refined.

Click the [Finish] button to create the child grid.
### 8.1.4 Extract subgrid

When working with large regional models you may find it difficult to manage as the associated file can become quite large. Additionally, you may be interested in doing further modeling - but only in a smaller area of the large regional model. Extracting a subgrid from your large regional model allows you to work with a smaller model in the area of interest.

You will find the option to Create a subgrid in the Numerical Workflow on the View/Edit Grid step.

![Image of the software interface showing the option to create a subgrid.](image)

Additionally you have the option to extract a subgrid from an existing grid (which has been run) by right clicking on the existing grid in the Model Explorer and selecting Extract subgrid...

By extracting a subgrid - all existing inputs (for example: inactive cells, properties, boundary conditions, observation wells, zone budget zones, and particles) that are in the existing grid will be transferred to the subgrid. Outputs are not transferred.
Visual MODFLOW Flex only supports extracting subgrids from Finite Difference grids. USG and Finite Element grids do not support extracted subgrids.

The model on the grid you are trying to extract a subgrid from must have been translated and run to completion and only the following engines are supported:

- MODFLOW 2000
- MODFLOW 2005
- MODFLOW-NWT

The following dialog appears when selecting Create subgrid...
By default a Subgrid name will be provided but you can adjust this.

You can adjust the Row start and end as well as the Column start and end to define your subgrid.

It may be helpful to load objects (from the Data and Model Explorer) to help you define your subgrid - simply select the appropriate checkbox and they will be displayed in the subgrid preview in this dialog - for example a River has been added to the subgrid preview in the above image.

Currently all Layers will be included in your subgrid. In future versions you will be able to select a subset of layers to include in your subgrid.

You can highlight the dry cells in your model display in the subgrid preview. This can be helpful to ensure that no dry cells land on the boundary of your subgrid.

Extracting Subgrids with Dry Cells

Please be aware that the boundary of your subgrid cannot fall on cells that...
are dry (from the regional model run). You must adjust the extents of your subgrid to ensure no dry cells fall on the boundary.

The toolbar at the top of the subgrid preview provides you with several options:

- Zoom In
- Zoom Out
- Zoom to Box
- Zoom to Full Extent
- Pan

Once you have defined your subgrid extents and select OK you will find the new subgrid (with corresponding model) will appear in the Model Explorer as well as a new Numerical Workflow will open. Select the subgrid in the Model Explorer to have it display in the new Numerical Workflow.

As mentioned previously, all inputs from the original grid will be transferred to your subgrid. Additionally, the subgrid will contain constant heads all the way around the perimeter with assigned heads values from the original model. From here you can continue to develop your new model based on your subgrid extraction.
8.2 Define Finite Element Mesh

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Create the superelement mesh, define the mesh size, and define vertical layering type and resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites:</td>
<td>The Conceptual Model has been defined.</td>
</tr>
<tr>
<td>Result:</td>
<td>A finite element mesh is created</td>
</tr>
<tr>
<td>Next Steps:</td>
<td>Convert Conceptual Model to FEFLOW finite element model</td>
</tr>
</tbody>
</table>

From the "Select Grid Type" step in the Conceptual Model workflow, click on the "Define Finite Element Mesh button":

The Define Finite Element Mesh wizard will appear on your screen, where you can configure the properties of the finite element mesh.
Specify a unique name for the finite element mesh in the Name text field.

**Defining the Superelement Mesh**

The Superelement Mesh represents the main geometry (points and segments) of the model region from which finite elements are generated. A superelement mesh is defined using the model boundary polygon geometry and, optionally, one or more “add-ins”. Add-ins are lines, points or polygons within the model boundary which Visual MODFLOW Flex uses as focal points to create nodes during finite element mesh creation.

By default, the Add-ins List contains the model boundary and any linear or point boundary condition currently defined for the conceptual model. Additional add-ins may be added to the list using data objects from the Data Explorer.

---

**Unstable Mesh Designs**

To avoid unstable mesh designs, it is recommended that all add-in objects are pre-processed such that the following conditions are met:
there is equal spacing between vertices along polylines and the edges of polygons,
- polygons in the same layer do not overlap or contain slivers, and
- vertices that appear to share common coordinates share them exactly

Data object geometry can be edited using the 2D Viewer editing tools. For more information, please see "Digitizing & Editing Geometry in 2D Viewers".

To add an Add-In features using a data object from the Data Explorer,

- Leaving the Define Finite Element Mesh window open, select the desired data object from the Data Explorer.
- In the Define Finite Element Mesh window, select the Add-in Lines/Points/Polygons button, located below the Add-ins List.

An Add-in may be included or excluded in the mesh creation, by checking or unchecking the corresponding check box, respectively. When an add-in is “checked” it will also be displayed in the adjacent 2D Viewer preview window.

Defining the Horizontal Mesh Settings

In the second dialog in the Define Finite Element Mesh wizard, you can define various discretization settings for the horizontal mesh.
Finite element meshes are created using the Triangle mesh generator, developed by J.R. Shewchuk (1970). Triangle provides various options for generating finite element meshes. These options are described briefly below.

Please Note: the switch letters in parenthesis beside the name of each option refers to the command line switches used by the Triangle mesh generator.

**Delaunay Triangulation Method (L-switch)**

Delaunay triangulation methods are typically used in finite element mesh generation as they tend to maximize the minimum angle of all the angles of the triangles in the triangulation, providing a nice set of triangles, and avoiding narrow “skinny” triangles. There are three triangulation methods from which to choose from: Constrained, Constrained Conforming and Conforming.

- **Constrained**: Triangulation in which each segment appears as a single edge in the triangulation. As such, segments are not subdivided, and new vertices are not added to the vertex set. A constrained Delaunay triangulation is not truly a Delaunay triangulation, because some of its triangles might not be Delaunay.
- **Constrained Conforming**: Triangulation in which triangles are constrained Delaunay; however, additional vertices may be added to the vertex set and segments may be subdivided to ensure a user-defined Minimum Angle constraint is satisfied. If a
minimum angle is not specified, vertices are added to ensure all angles are between 20 and 140 degrees.

- **Conforming**: Triangulation in which each triangle is truly Delaunay, and not just constrained Delaunay. Additional vertices may be added to the vertex set to enforce the Delaunay property.

**Meshing Algorithm (I-switch)**

Two Delaunay algorithms are provided for generating the finite element mesh: Divide and Conquer, and Incremental. Typically, the divide and conquer algorithm is preferred. However, if this algorithm fails, use the incremental algorithm.

**Total Number of Elements (Approx)**

For the Conforming and Constrained Conforming triangulation methods, you can specify the desired number of elements that comprise the finite element mesh. Please note that the specified number of elements cannot be less than the default number generated by Triangle, i.e., the number generated if this option is disabled.

**Minimum Angle**

For the Constrained Conforming triangulation method, a minimum angle can be specified. The specified angle will replace the default bound on the minimum angle (20 degrees). The specified angle may include a decimal point, but cannot be expressed in exponential notation.

**Refinement Options**

- **Edges of triangles along model boundary should have approx length**: Use this option to set the approximate length of edges (segments) that comprise the model boundary domain. Vertices will be added along the boundary, creating subsegments with the specified length. This option will refine the areas along the model domain boundary.

- **Edges of triangles along line should have approx length**: Use this option to set the approximate length of edges of triangles along line add-ins. Vertices will be added along the lines, creating subsegments with a specified length. The option will refine areas around line add-ins.

- **Refinement around point add-ins**: Use this option to refine areas around point add-ins. Refinement for point add-ins is defined by specifying the number of triangles directly around the points, and the desired distance from the point to the new vertices. Use the Gradation slider bar to specify the smoothness of the transition from the fine elements around the points to the coarser elements. A smoother transition will result in more elements, but will lead to more regular elements therefore improved model stability.

**Polygon Refinement**

You can use one or more polygon data objects that have been included in the superelement mesh (see "Create Finite Element Mesh") to define localized areas of mesh refinement. To do so, click the Polygons Refinement... button. The Polygon Refinement screen will appear.
In the left side of the Polygon Refinement window, all features in the polygon add-ins are listed in a table under the Polygon ID column. When a feature is selected, it is highlighted yellow in the adjacent 2D Viewer. Select the polygon feature that represents that area of the mesh for which refinement is to be performed. In the adjacent field under the Number of Elements column, enter the desired number of element that should exist in polygon area. Visual MODFLOW Flex will automatically refine the underlying triangles to equal the prescribed number of elements for the polygon area.

Once you have defined the refinement for the polygon features, click the [OK] button to apply the settings.

**Generating the Horizontal Mesh**

Once the above settings have been defined, click the [Generate] button to generate the horizontal mesh. Once generated, the Triangle output results, e.g., number of mesh vertices, triangles, edges, etc., are displayed in the text box (above the Generate button). These results may be copied to the clipboard by right-clicking anywhere inside the box, and selecting Copy to Clipboard from the pop-up menu.

The generated mesh will also display in the adjacent 2D Viewer window.

If you are not satisfied with the mesh, you can modify the settings and regenerate the mesh by selecting the Generate button again.
If you have selected polygon refinement, the mesh may look something similar to below.

---

**Defining Slice Elevations**

The third step in the finite element mesh creation involves defining slice elevations. Various settings are available for defining the slice elevations in the third dialog of the finite element mesh creation wizard (shown below).
The options available for defining slice elevations are similar to those for defining slice elevations for finite difference grids. For information on these settings, e.g., grid types, min layer thickness and layer refinement, please refer to "Defining the Vertical Grid".

Once the vertical mesh settings have been defined, click the [Finish] button to generate the finite element mesh. The finite element mesh will now appear in the Model Explorer tree under the Model Domain node as shown below, where it can be displayed in a 2D or 3D Viewer window, and used in Translation to Finite Element Model.
8.3 Define Unstructured Voronoi Grid

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Define add-ins for the grid, define the approximate number of cells, and the refinement options.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites</td>
<td>The Conceptual Model has been defined.</td>
</tr>
<tr>
<td>Result</td>
<td>An UnStructured Voronoi Grid is created and added to the Model Explorer tree.</td>
</tr>
<tr>
<td>Next Steps</td>
<td>Convert Conceptual Model to MODFLOW-USG</td>
</tr>
</tbody>
</table>

For more details on how MODFLOW-USG differs from traditional versions of MODFLOW, please see our website.
From the "Select Grid Type" step in the Conceptual Model workflow, click on the "Define UnStructured V-Grid" button:

The Define UnStructured V-Grid wizard will appear on your screen, where you can configure the inputs for the grid.
Specify a unique name for the UnStructured V-Grid in the Name text field.

**Defining the Grid Add-Ins**

The Add-ins for the grid represents the main geometry (points and line segments) of the model region from which Voronoi Cells are generated. The add-ins consist of the model boundary (a closed polygon) and, optionally, one or more “add-ins”. Add-ins are polylines, points, or polygons within the model boundary which Visual MODFLOW Flex uses as “node generators” for the grid generator. By default, the Add-ins List contains the model boundary and any linear or point boundary conditions defined for the conceptual model. Additional add-ins may be added to the list using data objects from the Data Explorer. Boundary conditions that are created with polygons are not included in the Add-ins, by default. (Generally, it is not recommended to include polygon-based boundary conditions that cover the entire model, as grid add-ins; for example, recharge or evapotranspiration. However, smaller polygons, for example those representing lakes, ponds, drains, general heads, etc., may be used)
To avoid unstable grid designs, it is strongly recommended that all add-in objects are pre-processed such that the following conditions are met:

- there is equal spacing between vertices along polylines and the edges of polygons,
- polygons in the same layer do not overlap or contain slivers, and
- vertices that appear to share common coordinates share them exactly.

Data object geometry can be edited using the 2D Viewer editing tools. For more information, please see "Digitizing & Editing Geometry in 2D Viewers".

To add an Add-In feature using a data object from the Data Explorer:

- Select the desired data object from the Data Explorer (while leaving the Define UnStructured Grid window open)
- In the Define unstructured grid window, select the [Add-in Lines/Points/Polygons] button, located below the Add-ins List.

An Add-in may be included or excluded in the grid creation, by checking or unchecking the corresponding check box, respectively. When an add-in is “checked” it will also be displayed in the adjacent 2D Viewer preview window.

When you are finished, click the [Next>>] button to proceed.

Defining the Grid Size and Refinement Settings

In the second dialog, you can define various settings that control the cell sizes and the approximate number of cells.
Unstructured grids are created using the Triangle mesh generator, developed by J.R. Shewchuk (1970). Triangle provides various options for generating unstructured grids. These options are described briefly below. Please note that the switch letters in parenthesis beside the name of each option refers to the command line switches used by the Triangle mesh generator. More details on “Triangle” are available on the Triangle website.

In Visual MODFLOW Flex, this mesh generator is used to create a Delaunay Triangulation, and from this, Voronoi Grid cells are created, which are suitable grids for MODFLOW-USG. More details on Voronoi Polygons and grids can be found on our website.

**Delaunay Triangulation Method (L-switch)**
Currently this option is always set to *Force all triangles to be Delaunay*.

**Meshing Algorithm (I-switch)**
Two Delaunay algorithms are provided: *Divide and Conquer*, and *Incremental*. Typically, the divide and conquer algorithm is preferred. However, if this algorithm fails, use the incremental algorithm.
**Total Number of Elements (Approx)**
This setting is a target for the grid generator, and is an approximation of how many cells will comprise the areas outside the refinement regions. The actual total number of cells generated in the grid will depend on the geometry add-ins you have included, and the refinement settings, and will typically be much higher. For example:

- In the figure above there are 2 wells, and 3 polyline boundaries. Using the default refinement settings and the number of elements as 1000 results in approximately 4300 unstructured grid cells (per layer).
- For the same example, using 10000 as the total number of elements, 100 for the edge length around lines, and 25 as the spacing from points to new nodes, and middle-range Gradation, results in 19000 unstructured grid cells (per layer).

As a rule-of-thumb, the actual number of Voronoi cells that will result from the grid generator is typically within the same order of magnitude as the value you define for the "Total Number of Elements".

Fortunately in Visual MODFLOW Flex, you can repeat the grid generator several times, create multiple grids, and find the most suitable grid size and refinement level for your project.

**Quality Mesh (Minimum Angle)**
The default is 30 degrees for creating Voronoi polygons, and should be suitable in most cases. However, if you encounter difficulties with the default value, you may experiment with values of 15 or 20 degrees. In theory, values with lower angles may work, though we have limited testing experience with these geometries.

**Refinement Options**
The refinement options are separated into three categories, each of which can have its own unique refinement levels:

- Superelement border edges: (the conceptual model boundary)
- Line add-ins (linear boundary conditions such as constant heads, rivers, drains, etc.)
- Point (well) add-ins

For each of the refinement settings, you enter an "edge length" (defined using the same unit convention as the project's units), and this will result in an "equivalent" approximate cell size.

- **Refinement around superelement border edges**: Use this option to set the approximate length of edges (segments) that comprise the model boundary domain. Vertices will be added along the boundary, creating subsegments with the specified length. This option will refine the areas along the model domain boundary.
- **Refinement around line add-ins**: Use this to set the approximate target cell size around the polyline add-ins. Vertices will be added along the lines, creating subsegments with a specified length. The option will refine areas around line add-ins.
- **Refinement around point/well add-ins**: Use this option to refine the grid around point add-ins. Refinement for point add-ins is defined by specifying the number of triangles directly around the points, and the desired distance from the point to the new vertices. The default of 10 triangles should be adequate for most cases; in order to adjust the cell size around the pumping wells, you need to change the "Space from points to new nodes" value; a smaller value will result in smaller cell sizes.

- **Gradation**: Use the Gradation slider bar to specify the smoothness of the transition of the cell sizes from around the refinement regions to the more coarser (background) cell sizes. A smoother transition will result in more cells, but will lead to a more gradual change in cell size, which is recommended for MODFLOW-USG grids.

**Layer Refinement**

You can refine the conceptual layers of the model into sub-layers using the Layer Refinement Settings button, which opens a window with a table and allows you to specify the number of evenly divided sub-layers for each conceptual layer in your model.

![Unstructured Grid Layer Refinement](image)

**Generating the Grid**

Once the above settings have been defined, click the [Generate] button to generate the unstructured grid. Once generated, the Triangle output results, e.g., number of mesh vertices, triangles, edges, etc, are displayed in the text box (above the Generate button). These results may be copied to the clipboard by right-clicking anywhere inside the box, and selecting Copy to Clipboard from the pop-up menu.
The generated grid will also display in the adjacent 3D Viewer window. If you are not satisfied with the grid, you can modify the settings and regenerate the grid by selecting the [Generate] button again.

Once the grid is defined, click the [Finish] button to complete the process. The UnStructured V-Grid will now appear in the Model Explorer tree under the Model Domain node as shown below, where it can be displayed in a 3D Viewer window.
Next you are ready to generate a numerical model from this UnStructured Grid. This is explained in the section **Converting to MODFLOW-USG**.

When you are done previewing the grid, click on the Next button on the workflow toolbar.

### 8.3.1 Cell Numbering

The order in which cells are numbered for UnStructured Voronoi grids is as follows.

1. CellID 1 starts at the first vertex of the model boundary polygon in the uppermost layer and works along the edge (typically in a counter-clockwise direction
2. After this, the add-ins are numbered, starting with polylines, polygons. Vertices of the shapes are numbered based on their order:

3. Then point features (including wells) are added:
4. After this, the remaining cells are numbered, filling in the gaps, starting with layer 1, and working downwards.

8.4 Define Unstructured QuadTree Grid

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Define add-ins for the grid, define the approximate number of cells, and the refinement options.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites</td>
<td>The Conceptual Model has been defined.</td>
</tr>
<tr>
<td>Result</td>
<td>An UnStructured Q-Grid is created and added to the Model Explorer tree.</td>
</tr>
<tr>
<td>Next Steps</td>
<td>Convert Conceptual Model to MODFLOW-USG</td>
</tr>
</tbody>
</table>
Quadtree grids are based on rectangular-shaped grid cells that can be sequentially subdivided into four equally-sized cells. Quad-Based grid refinement is a simple way to focus resolution in areas of interest in order to better represent hydraulic gradients or to more accurately represent known variations in hydraulic properties or boundary conditions. Examples of Quad-Based grids include Nested grids, Quadtree grids, and Octree grids (the case when a Quadtree grid is refined in three dimensions).

![Quadtree Grid](image)

(a) Quadtree Grid

![Smoothed Quadtree Grid](image)

(b) Smoothed Quadtree Grid

![Nested Grid](image)

(c) Nested Grid

![Octree Grid](image)

(d) Octree Grid

Images adapted from Panday (2013)

In order to reduce numerical errors and improve accuracy of the simulation, it is generally recommended that Quad-Based grids are smoothed so that every cell is connected to no more than two cells in any direction. Doing so can simplify the calculation of the Ghost Node locations (for the Ghost Node Correction [GNC] package). Also, keeping with a limit of two connections per cell face may yield minimal errors in the calculated heads/fluxes such that the GNC package may not be needed. For more details on how MODFLOW-USG differs from traditional versions of MODFLOW, please see our website.

From the "Select Grid Type" step in the Conceptual Model workflow, click on the "Define UnStructured Q-Grid" button:
The Define UnStructured Q-Grid wizard will appear on your screen, where you can configure the inputs for the grid. There are five groups of controls in the wizard:

1. Toolbar
2. Grid Size
3. Layer Refinement
4. Feature Refinement
5. Viewer and Statistics
Toolbar

The toolbar includes the following controls:

<table>
<thead>
<tr>
<th>Name</th>
<th>Q-Grid1</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Reset Refine" /></td>
<td>resets the grid to the Grid Size settings and removes all feature refinement and smooth operations</td>
</tr>
<tr>
<td><img src="image" alt="Smooth" /></td>
<td>refines cells that have four or more neighbors along the same cell face</td>
</tr>
<tr>
<td><img src="image" alt="Add Selected Feature" /></td>
<td>adds a feature selected in the Data Explorer to the refine Feature Refinement control</td>
</tr>
<tr>
<td><img src="image" alt="Delete Selected Feature" /></td>
<td>deletes the selected feature from the Feature Refinement control</td>
</tr>
<tr>
<td><img src="image" alt="Settings" /></td>
<td>provides visualization settings for the selected feature in the Viewer</td>
</tr>
</tbody>
</table>

Defining the Grid Size

By default, Visual MODFLOW Flex discretizes the Q-grid layers into 32 rows and 32 columns. The Columns and Rows fields allow you to define the base grid size. You can also specify the Cell Height and Cell Width; the Rows and Columns (and potentially the extents) will be updated accordingly.
Defining the Vertical Grid

The Layer Refinement controls allow you to subdivide the conceptual layers into the specified sublayers. All layers will be deformed. Pinchouts

<table>
<thead>
<tr>
<th>Layer Name</th>
<th>Layer Refinement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layer 1</td>
<td>1</td>
</tr>
<tr>
<td>Layer 2</td>
<td>1</td>
</tr>
<tr>
<td>Layer 3</td>
<td>1</td>
</tr>
<tr>
<td>Layer 4</td>
<td>1</td>
</tr>
</tbody>
</table>

Defining the Grid Size and Refinement Settings

Features can be refined at locations where the grid intersects selected polygons, polylines, points or well objects from the Conceptual Model Boundary Conditions and Data Explorer objects.

By default, all of the boundary conditions assigned in the conceptual model are included in the Refine Features table. Polygons that are added to the table, also have their (polyline) boundaries added so that you can refine the edge of the polygon and/or its interior. Additional features can be added from the Data Explorer to the table using the Add Selected Feature button.

To refine the grid around a selected feature, click the Refine button. You can click the Refine button several times to refine the grid sequentially. You can also set a threshold value for the Min Area field of a given feature (in project length-squared units). When a non-zero value is set, you can sequentially refine the model until refining operation(s) would lead to grid cells smaller than this threshold area. Once a non-zero value has been specified, you can also automatically refine the grid near the selected feature by clicking the...
button which iteratively refines all intersecting cells until a subdivision operation would cause a grid cell area to be less than the threshold Min Area value. The Min Area value applies to both the the Refine and the Refine to Min buttons.

**Please Note:** The automated grid refinement using the Refine to Min button can be fairly slow for operations involving large grids, features with many vertices, and/or low specified Min Area values and may require some time to complete as each refinement step will quadruple the number of cells that intersect the selected feature(s). You can stop the automated refinement process part-way through the operation and it will complete the current iteration by clicking the stop button.

The Refine Features table also contains a Visible field which toggles the visibility of the features in the Preview window below the table and a Count field which indicates how many times the feature has been refined.

**Q-Grid Preview**

A preview of the Q-Grid will be displayed in the 3D Viewer window in the lower right quadrant of the Defined Q-Grid window. If you are not satisfied with the grid, you can modify the settings and regenerate the grid by selecting the Reset Refine button. A status bar can be found at the bottom of the preview window that indicates the total number of grid cells based on the current settings as well as the Maximum neighbor length ratio. This ratio indicates the largest relative difference in length or width between adjacent connected cells. As discussed above, a lower ratio may reduce errors in solving the flow equation and a value of 2 is generally ideal. You can refine cells that are four or more times wider/longer than a neighbor by clicking the smooth button. If the adjacency ratio is larger than 4, you may need to smooth the grid several times.
Finishing the Design

Once the grid is defined, click the [Finish] button to complete the process. The UnStructured Q-Grid will now appear in the Model Explorer tree under the Model Domain node as shown below, where it can be displayed in a 3D Viewer window.
Next you are ready to generate a numerical model from this UnStructured Q-Grid. This is explained in the section [Converting to MODFLOW-USG](#).

When you are done previewing the grid, click on the Next button on the workflow toolbar.
9 Converting Conceptual Models to Numerical Models

Visual MODFLOW Flex allows you to translate your conceptual model to one or more numerical models that will run in a groundwater modeling engine. Finite difference and finite control volume models can be run and analyzed within the Visual MODFLOW Flex environment, while finite element models are exported to run in FEFLOW. During translation, Visual MODFLOW Flex automatically populates the specified grid or mesh with the defined geological formations, boundary conditions, and property zone attributes, and creates the necessary input files for loading into your desired simulator.

Separate workflows will be generated every time a conceptual model is converted into a numerical model. Structured finite difference grids (which work with the following engines: MODFLOW-2000, -2005, -NWT, -LGR, -SURFACT, SEAWAT, MT3DMS, and RT3D) are converted to the numerical workflow, while unstructured finite control volume grids are converted to the unstructured grid USG workflow. In each workflow, you can view/edit the properties and boundaries on the numerical grid, then translate and run a groundwater modeling engine and view the results.

For finite element models, Visual MODFLOW Flex will generate the .FEM problem file for loading into FEFLOW.

⚠️ Please Note: FEFLOW v.5.4 or later is required for opening and running FEFLOW models.

This chapter walks you through the steps involved in translating a conceptual model to a numerical model, and includes information on the following topics:

- Converting to MODFLOW-Finite Difference
- Converting to MODFLOW-USG
- Converting to FEFLOW

9.1 Converting to MODFLOW-2000/2005

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Create a MODFLOW numerical model by populating the selected grid with data from the conceptual model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites:</td>
<td>A Finite Difference Grid has been created.</td>
</tr>
<tr>
<td>Result:</td>
<td>A numerical model is generated and added to the Model Explorer tree. The numerical model workflow will appear.</td>
</tr>
<tr>
<td>Next Steps:</td>
<td>Numerical Modeling Workflow</td>
</tr>
</tbody>
</table>
Once you have the conceptual model designed and at least one numerical grid, you are ready to populate this grid with the conceptual data.

Proceed to the "Convert to Numerical Model" step in the Conceptual Model workflow; this is shown in the display below.

The Select Grid combo box will list all the finite difference grids you have created for your project, including grids for use in Local Grid Refinement (LGR).

Select the grid you wish to use, and click on the "Convert to Numerical Model" button. (Alternatively, you can select the desired numerical grid from the Model Explorer tree, right mouse click, and select "Convert to Numerical Model from the pop-up menu.)

After clicking on the conversion button, a new numerical model workflow window will appear which includes the steps for the numerical model. In the first window, you will see the progress of the conceptual to numerical conversion. This conversion could take several minutes, depending on the size and type of grid you used, and the complexity of the conceptual model inputs.

During the conversion, the run node will be added to the model explorer under the numerical grid that you selected. This tree will be populated with all the inputs from your conceptual model: Properties, Boundary Conditions, Wells, etc. A grid may have multiple run nodes to facilitate simulating multiple scenarios, in the PRO and Premium versions of Visual MODFLOW Flex. For more information on working with multiple model scenarios in a project, please see the section on Working with Multiple Model Scenarios.
When the conversion is complete, you can click on the Next button on the workflow toolbar, which will take you to the Define Properties. You can then proceed through the numerical model workflow.

Converting MODFLOW Grids with Child Grids

When you convert a Grid that contains one or more child grids (for an LGR run), the inputs on the Model Explorer tree will contain objects for the parent model and each child model.

In the example below, Conductivity1, Storage1, InitialHeads1 are the property values for the Parent Grid and Conductivity-Child grid1, Storage - Child grid1, and InitialHeads - Child grid 1 are the property values for ChildGrid1.

The same convention applies for boundary condition cells: there is one entry for the parent grid boundary cells and one entry for each of the child grid cells.
After a successful MODFLOW-LGR run, you will see multiple entries in the Output directory: Heads for the parent grid and Heads for each of the child grids, and likewise for Drawdown.
9.2 Converting to MODFLOW-USG

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Create a MODFLOW-USG numerical model by populating the selected grid with data from the conceptual model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites:</td>
<td>An UnStructured V-Grid or Q-Grid has been created.</td>
</tr>
</tbody>
</table>
Once you have the conceptual model designed and at least one UnStructured V-Grid or Q-Grid, you are ready to populate this grid with the conceptual data.

Proceed to the "Convert to MODFLOW-USG Model" step in the Conceptual Model workflow; this is shown in the display below.

The Select Grid combo box will list all the UnStructured grids you have created for your project.

Select the grid you wish to use, and click on the "Convert to Numerical Model" button.
Conversion times!

This conversion process could take several minutes, depending on the number of cells in the grid, the number of layers, and the complexity of the conceptual model inputs. **Please be patient.**

After the conversion is complete, a new numerical model workflow window will appear which includes the steps for viewing and editing the UnStructured grid numerical model.

During the conversion, a "Run" node will be added to the Model Explorer under the UnStructured grid that you selected. This Model Explorer tree will be populated with all the inputs from your conceptual model: Properties, Boundary Conditions, Wells, etc.
When the conversion is complete, you can click on the Next button on the workflow toolbar, which will take you to the View/Edit Properties. You can then proceed through the MODFLOW-USG numerical model workflow.

9.3 Converting to FEFLOW

Quick Overview

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Create a FEFLOW .FEM file by populating the selected mesh with data from the conceptual model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pre-requisites</td>
<td>A Finite Element Mesh has been created</td>
</tr>
</tbody>
</table>
Once you have the conceptual model designed and at least one finite element mesh, you are ready to populate this mesh with the conceptual data.

Proceed to the “Translate to Finite Element Model” step in the Conceptual Model workflow; this is shown in the display below.

### Translate to Finite Element Model Options

Click on the Translate to Finite Element model button, and the following dialog will appear. (Alternatively, you can select the Conceptual model folder in the Model Explorer tree, and select Translate to Finite Element Model... from the pop-up menu.)
Project Description
A brief description of the project. By default, this is the name of the current Visual MODFLOW Flex project.

Finite Element Mesh
Select the desired finite element mesh from the combo box to use for model translation.

Output Name
Click the button and specify the output name and directory for the *.FEM file that is created during translation.

The .FEM file is an ASCII file containing information on the problem class and model properties. This file is required for importing your model into FEFLOW.

The default file name is “Project_name.FEM” and the default directory on your computer is [Project Folder]\Numerical Models.
Translation Log File
When a numerical model is translated in Visual MODFLOW Flex, a log file is automatically generated and saved on your computer. By default the log file name is [Project_Name].LOG and the directory is [Project Name]/Numerical Models.

Click the button to specify a new file name and directory.

Problem Class
Currently, Visual MODFLOW Flex only supports the separate flow problem class.

Simulation Type
Select Steady State or Transient from the combo box.

If the Steady State option is selected, Visual MODFLOW Flex will prepare the data set for a steady-state flow simulation, and will automatically use the data from the first stress period of each boundary condition and pumping well defined in your conceptual model.

If the Transient Flow option is selected, Visual MODFLOW Flex will automatically merge all the different time period data defined for each pumping well and boundary condition into the stress period format required by FEFLOW.

Flow Type
Select the flow type of the problem class. Choose from the following options: Saturated media (groundwater), unsaturated media, Unsaturated steady-state linearized Richards equation.

Translation Format
The output .FEM file generated by Visual MODFLOW Flex during translation is currently compatible with FEFLOW v.5.3.11 or later.

Start Date
The default start date is the date specified in the conceptual model settings. The Start Date of the model is the date corresponding to the beginning of the simulation. This date is relevant only for transient flow simulations where recorded field data may be used for defining time schedules for selected boundary conditions.

Start Time
Specify the simulation start time. The Start Time of the model is the time corresponding to the beginning of the simulation. This time is relevant only for transient flow simulations where recorded data may be imported for defining time schedules for selected boundary conditions.
**Steady-State Simulation Time**

A steady state simulation time is required for Steady State Flow simulation. This parameter is not used if you have selected Transient Flow. Although the simulation will always be run to the same equilibrium solution in Steady State, the total amount of water passing through boundary conditions (i.e. the cumulative value of the solution) depends on the amount of time simulated.

The Packages frame contains a list of existing boundary conditions for the conceptual model being translated. Here you can include/exclude boundary conditions by selecting the appropriate check box in the Translate column. The corresponding FEFLOW condition type is listed for each boundary condition under the FEFLOW Conditions column.

You can also select how Recharge and Evapotranspiration should be represented: As Flow Properties or as Type2 Boundary Condition.

Click the [Next] button to initiate the model translation. During translation, output details are displayed in the translation log. Once finished, click the [Finish] button to exit the translation wizard.
10 Numerical Modeling Workflow - Finite Difference Grids

This chapter presents information on editing data in a numerical model workflow for regular grids; the numerical model can originate from:

- Importing a Visual MODFLOW Classic or MODFLOW project, or
- Executing a Conceptual to Numerical Model conversion

The numerical modeling workflow provides the tools for building the numerical model (properties and boundaries assigned to grid cells), running the MODFLOW engines, and analyzing the results. For information on editing data in an unstructured grid, see the related section on the numerical model workflow for unstructured grids.

The following sections are covered:

1. Define Modeling Objectives
2. Define/Edit Grid (if importing or creating a new numerical model)
3. View/Edit Properties (geometry and parameter edits)
4. View/Edit Boundary Conditions (constant heads, drains, pumping wells, recharge, etc.)
5. Define Observation Wells
6. Define Zone Budget Zones
7. Define Particles
8. Translation
9. Run Numerical Engines
10. View Results
    - Contours and Color Floods
    - Charts
Before You Start!

If you need to create, modify, or maintain a model that utilizes any of the following features, you must continue to use Visual MODFLOW Classic interface for this:

- Flow Engines (MODFLOW-96)
- Transport Engines (MT3D99, PHT3D)
- Packages (ETS1, STR, MNW)
- MGO

10.1 Define Objectives

The first step in the workflow for developing and running numerical models in Visual MODFLOW Flex is to define the modeling objectives. This entails selecting the desired flow and transport simulation options. The selected combination of flow options (e.g. saturated groundwater flow) and transport options (e.g. reaction and sorption models) will narrow the list of available flow and transport engines and generate associated input variables (including units and default values). The purpose of this step is to develop the necessary inputs required to build the model based on your understanding of the relevant physical processes present within the study area.

The numerical workflow supports the following engines:

<table>
<thead>
<tr>
<th>Engines supported in Visual MODFLOW Flex</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Flow</strong></td>
</tr>
<tr>
<td>MODFLOW-SURFACT¹</td>
</tr>
<tr>
<td>SEAWAT²</td>
</tr>
</tbody>
</table>
Notes:
1 - MODFLOW-SURFACT is a commercial groundwater modeling engine that supports coupled groundwater flow and transport simulations.
2 - SEAWAT is a coupled variable density groundwater flow and transport model.
3 - MOD-PATH3DU is a particle-tracking algorithm developed by a collaboration between S.S. Papadopulos & Associates (SSPA) and the University of Waterloo. It is not installed directly with Visual MODFLOW Flex and must be downloaded and installed separately from the mod-PATH3DU website.
Understanding Default Values at the Modeling Objectives Step

Values set at the Define Modeling Objectives workflow step are used by Visual MODFLOW Flex to populate the applicable arrays for the flow and transport model consistent with the modeling objectives upon reaching the Define Properties Step for the first time. Constant values (discussed below) are the exception to this and are discussed below.

Once the applicable arrays have been created in the Model Explorer, changes to the default values at the Define Modeling Objectives will not be reflected in the Flow and Transport Models. To reset values for a group of properties (e.g., Flow/Conductivity or Transport/Initial Concentrations) delete the relevant node from the Model Explorer Tree, update the default value at the Modeling Objectives step and proceed to the Define Property Zones step to regenerate the deleted arrays with the updated defaults.

Constant values that are not saved to distributed grid arrays can be changed at any time at the Modeling Objectives Step and subsequent changes will have be reflected in the applicable translated flow/transport models. Examples include:

- **Species Parameters:**
  - Reference Concentrations and the density and viscosity slopes for SEAWAT models

- **Model Parameters:**
  - Stoichiometric Yield coefficients for RT3D reactions

Flow Objectives

Under the left panel of Define Modeling Objectives, you can specify a Start Date for the project. Pumping well data and calibration data (head and concentration observation wells) are entered and save as absolute (calendar) dates. Therefore, it is important to choose a relevant start date at this step that coincides with the start time of your simulation.

Flow and Simulation Type

There are several flow condition options available for flow modeling in the numerical workflow:
Flow Engines and Modeling Objectives Supported in Visual MODFLOW Flex

<table>
<thead>
<tr>
<th>Simulation Type</th>
<th>Saturated Flow</th>
<th>Variably Saturated Flow</th>
<th>Zones</th>
<th>MODPATH</th>
<th>MODPATH3DU</th>
<th>Transport</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Constant Density</td>
<td>Variable Density</td>
<td>Groundwater</td>
<td>Vapor Flow</td>
<td>Budget</td>
<td></td>
</tr>
<tr>
<td>MF-2000</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>MF-2005</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>MF-NWT</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>MF-LGR</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>SEAWAT</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>MF-SURFACT</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>

Notes:
1 - Abbreviations for variably saturated flow processes in the Vadose Zone:
- UZF = unsaturated zone flow package is based on 1-D unsaturated flow of a diffusion wave pseudo soil
- V = Van Genuchten relative permeability and saturation relationship
- BC = Brooks-Corey relative permeability and saturation relationship

Under Property Settings, you can modify the default flow parameters.

⚠️ Start Date

The start date will be used to retrieve pumping well and head/concentration observation data for the model run. When you define well data with absolute (calendar) dates, it is important that your start date reflects the actual start time for the model run. The well data must fall on or after that start date. Otherwise, these data will not be included in the simulation.

Also the start date cannot be changed once it has been set. If you inadvertently set the wrong start date, you can import your pumping well data and observation data in relative times (e.g. starting at 0), and you will see no difference in the numerical model inputs/outputs.
Start Date

The model start date for this exercise should be set to 1/1/2000. Visual MODFLOW Flex uses a standard Windows date picker; a few tips are shown below on how to use this. Click on the button shown below, to load the Windows date picker.

The standard Windows calendar will appear. Click on the month in the header (as shown below):

All months for the current year will appear as shown below. Click on the year in the header:
A range of years will then appear as shown below. Click on the range of years in the header:

A list of years for the previous decade will appear. You can then use the < or > buttons to change the year. Once you have reached the desired year (2000 for this example), select this on the calendar as shown below:

A list of months will then appear for that year. Select January for this example, as shown below.
Finally, select "1" from the calendar as shown below:

The selected date will then appear for the Start Date.

**Transport Objectives**

The transport objectives are defined on the right panel of the Define Modeling Objectives window. When you select the "Transport Active" check box at the top of the screen, then the options can be defined as explained and shown below.

**Retardation Model**

Transport engine retardation model options are currently supported in Visual MODFLOW Flex as follows:

<table>
<thead>
<tr>
<th>Description</th>
<th>MT3D</th>
<th>RT3D</th>
<th>SEAWAT</th>
<th>SURFA CT</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Sorption</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Linear sorption (equilibrium controlled)</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Freundlich (equilibrium-controlled)</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
</tr>
</tbody>
</table>
### Numerical Modeling Workflow - Finite Difference Grids

#### Reaction
Transport engine reaction model options are currently supported in Visual MODFLOW Flex as follows:

<table>
<thead>
<tr>
<th>Description</th>
<th>MT3D</th>
<th>RT3 D</th>
<th>SEAW AT</th>
<th>SURFA CT</th>
</tr>
</thead>
<tbody>
<tr>
<td>No kinetic reactions</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
</tr>
<tr>
<td>First-order irreversible decay</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
<td>☑</td>
</tr>
<tr>
<td>Zeroth-order irreversible decay</td>
<td>☑</td>
<td>☑</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Instantaneous aerobic degradation of BTEX</td>
<td></td>
<td>☑</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors

Rate-Limited Sorption

Double-Monod Model

Sequential First-Order Decay

Aerobic/Anaerobic PCE/TCE Dechlorination

Instantaneous biodegradation of BTEX

**Note:** MODFLOW-SURFACT supports additional reactions listed in the table above; however, these are not fully supported in the current version of Visual MODFLOW Flex.

The selections you make for Sorption and Reaction options will determine what parameters you need to define for your model. Default values can be defined in the "Species Parameters" and "Model Parameter" tabs. After the model is created, you can define additional zonation for Transport Properties at the "Define Properties" step. Detailed information regarding these parameters can be found in the MT3DMS, RT3D, SEAWAT, and SURFACT Users Manuals.

The New Species and Delete Species buttons allow you to add and remove species to your Transport simulation. This option is available only for certain transport engines.

When you Species Parameter tab is selected, use the add button (➕) to add a new species, or use the remove button (✖️) to remove the selected species.

ℹ️ **Please Note:** the remove button will remove the last species in the list.

The reaction model parameters window is shown below. For more information, see the individual sections describing [Species Parameters](#) and [Model Parameters](#).
When you are finished, click (Next Step) to proceed.

10.1.1 Project Property Settings (Flow)

The default flow parameters are set in project property settings affecting can are set in the Default Project Property Settings area which is divided into four groups:

Values will serve as the defaults for the model when the grid arrays in the model explorer are first populated. Note that once the grid arrays in the model explorer have been populated, changing the default values will not have any effect on the model. Changes can be made at the Define Properties step or to reset the values of a group back to defaults, delete the applicable node on the model explorer, update the default values as desired at the Modeling Objectives, and proceed to the Define Properties step to regenerate the deleted arrays.
**Conductivity**

- **Kx** - Hydraulic conductivity in the direction of the model X-axis, specified in conductivity units [L/T]
- **Ky** - Hydraulic conductivity in the direction of the model Y-axis, specified in conductivity units [L/T]
- **Kz** - Hydraulic conductivity in the direction of the model Z-axis, specified in conductivity units [L/T]

**Storage**

- **Ss** - specific storage, [-]
- **Sy** - specific yield, [-]

ℹ️ **Please Note**: Effective porosity (Ep) and total porosity (Tp) values are specified in the Project Settings window.

**Initial Heads**

- Initial heads, [L]

**Unsaturated Zone Properties** - UZF package only

- **EPS** - Brooks-Corey exponent $\varepsilon$, [-]
- **THTi** - initial water saturation $\theta_i$, [-]
- **THTs** - residual water saturation $\theta_r$, [-]
10.1.2 Species Parameters

The Species Parameters include the Sorption and Reaction parameters used by the selected transport settings. The available parameters will depend on what sorption and reaction settings you selected in the Modeling Objectives workflow step. The parameters presented in the Species Parameters Database window are from the parameters listed in the Species Parameters Tab in the Modeling Objectives workflow step.

If no sorption or reactions are selected in the current transport variant, then no sorption or reaction parameters are required for the simulation, and there will not be an option for "Species Parameters" at the Define Properties workflow step.

The parameters are described by retardation model and reaction model below and for the special case of variable density flow (SEAWAT) models.

**General**

The following species parameters will be available for any model where transport is a defined modeling objective:

- **Name**: the short name of each species to be modeled (e.g. TCE for trichloroethylene)
- **Description**: a more descriptive name for each species to be modeled (e.g. trichloroethylene when modeling TCE)
- **Initial Concentration**: the concentration of each species throughout the model domain at the start of the simulation [default=0]. See Theory.

**Retardation Model - Species Parameters**

The list of parameters required for each species based on each retardation model is provided below, associated base units are provided in square brackets [], where L signifies length (e.g. meters, feet), M signifies mass (e.g. kilograms, pounds), and T signifies time (e.g. seconds, days, years).

A tabulation of which retardation model is supported by each of the transport engines is provided in the Define Modeling Objectives workflow step. For more detailed documentation on each of the retardation model parameters, please refer to the documentation of the relevant transport engine (i.e. MT3DMS, RT3D, SEAWAT, or MODFLOW-SURFACT).

*Linear Isotherm (equilibrium controlled)*

- $K_d$ is the distribution coefficient $[1/(ML^3)]$

*Freundlich (equilibrium-controlled)*

- $K_f$ is the Freundlich constant $[1/(ML^3)^a]$
- $a$ is the Freundlich exponent [-]
Langmuir (equilibrium-controlled)
- $K_I$ is the Langmuir is constant \([1/(M/L^3)]\)
- $S$ is the total concentration of sorption sites available [-]

First order kinetic sorption (non-equilibrium)
- $K_d$ is the distribution coefficient \([1/(M/L^3)]\)
- $K_{mass}$ is the first-order kinetic mass transfer coefficient [1/T]

Dual-domain mass-transfer without sorption
- $S_{CONCIM}$ is the initial concentration in the distribution coefficient \([1/(M/L^3)]\)
- $K_{mass}$ is the first-order kinetic mass transfer coefficient [1/T]

Dual-domain mass-transfer with linear sorption in mobile domain
- $S_{CONCIM}$ is the initial concentration in the immobile domain \([M/L^3]\)
- $K_d$ is the distribution coefficient for the mobile domain \([1/(M/L^3)]\)
- $K_{mass}$ is the first-order kinetic mass transfer coefficient [1/T]

Dual-domain mass-transfer (with the same linear sorption in mobile and immobile domains)
- $S_{CONCIM}$ is the initial concentration in the immobile domain \([M/L^3]\)
- $K_d$ is the distribution coefficient for both the mobile and immobile domain \([1/(M/L^3)]\)
- $K_{mass}$ is the first-order kinetic mass transfer coefficient [1/T]

Dual-domain mass-transfer (with different linear sorption in mobile and immobile domains)
- $S_{CONCIM}$ is the initial concentration in the immobile domain \([M/L^3]\)
- $K_d$ is the distribution coefficient for the mobile domain \([1/(M/L^3)]\)
- $K_{mass}$ is the first-order kinetic mass transfer coefficient [1/T]
- $K_{dIm}$ is the distribution coefficient for the immobile domain \([1/(M/L^3)]\)

Dual-domain mass-transfer with nonlinear sorption in mobile domain
• **SCONCIM** is the initial concentration in the immobile domain [ML³]

• **Kf** is the Freundlich constant for the mobile domain [1/(ML³)⁰]

• **a** is the Freundlich exponent for the mobile domain [-]

• **K_mass** is the first-order kinetic mass transfer coefficient [1/T]

**Dual-domain mass-transfer (with the same nonlinear sorption in mobile and immobile domains)**

• **SCONCIM** is the initial concentration in the immobile domain [ML³]

• **Kf** is the Freundlich constant for both the mobile and immobile domain [1/(ML³)⁰]

• **a** is the Freundlich exponent for both the mobile and immobile domain [-]

• **K_mass** is the first-order kinetic mass transfer coefficient [1/T]

**Dual-domain mass-transfer (with different linear sorption in mobile and immobile domains)**

• **SCONCIM** is the initial concentration in the immobile domain [ML³]

• **Kf** is the Freundlich constant for the mobile domain [1/(ML³)⁰]

• **a** is the Freundlich exponent for the mobile domain [-]

• **K_mass** is the first-order kinetic mass transfer coefficient [1/T]

• **KfIm** is the Freundlich constant for the immobile domain [1/(ML³)aim]

• **aim** is the Freundlich exponent for the immobile domain [-]

⚠️ **Please Note:** Immobile species must be at the bottom of the species list on the Transport Modeling Objectives; be sure to add these to the species list last.

**Reaction Model - Species Parameters**

The list of parameters required for each species based on each reaction model is provided below, associated base units are provided in square brackets [ ], where L signifies length (e.g. meters, feet), M signifies mass (e.g. kilograms, pounds), and T signifies time (e.g. seconds, days, years).

A tabulation of which reaction model is supported by each of the transport engines is provided in the Define Modeling Objectives workflow step. For more detailed documentation on each
of the retardation model parameters, please refer to the documentation of the relevant transport engine (i.e. MT3DMS, RT3D, SEAWAT, or MODFLOW-SURFACT).

**Please Note:** The RT3D reactions have specific default species for each of its associated reaction modules. Selecting a reaction module specific to RT3D will reset the species list and all associated species and model parameters.

**First-Order Irreversible Decay**

- **K_mobile** is the first-order decay rate coefficient for dissolved constituents in both the mobile and immobile domain [1/T]
- **K_sorbed** is the first-order decay rate coefficient for sorbed constituents in both the mobile and immobile domain [1/T]

**Please Note:** First-order decay rate coefficients can be derived from half-life values, which are more commonly available. Concentrations that follow first-order decay can be shown to change over time by the equation:

\[
C_t = C_0 e^{-kt}
\]

where:

- \(C_t\) is the concentration of the constituent at time \(t\) [M/L^3]
- \(C_0\) is the initial concentration of the constituent [M/L^3]
- \(k\) is the first order decay rate [1/T]
- \(t\) is elapsed time [T]

When the constituent concentration reaches half of its initial concentration (i.e. \(C_t = 0.5 \times C_0\)), the equation above can be rewritten as:

\[
k = \ln 2 / t_{1/2}
\]

where:

- \(k\) is the first order decay rate [1/T]
- \(t_{1/2}\) is the half-life of the constituent \(C\) [T]

**Zeroth-Order Irreversible Decay**
• **K\(_{\text{mobile}}\)** is the zeroth-order decay rate coefficient for dissolved constituents in both the mobile and immobile domain [1/T]

• **K\(_{\text{sorbed}}\)** is the zeroth-order decay rate coefficient for sorbed constituents in both the mobile and immobile domain [1/T]

**RT3D Reaction Modules**

When selecting one of the reaction modules associated with RT3D:

• Instantaneous aerobic degradation of of BTEX;
• Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors;
• Rate-Limited Sorption;
• Double-Monod Model;
• Sequential First-Order Decay; or
• Aerobic/Anaerobic PCE/TCE Dechlorination

**RT3D Decaying Source**

• **Source Decay Constant** is the first-order decay rate coefficient for decaying sources [1/T]

RT3D uniquely supports specified "constant" concentration boundary conditions with a decaying source term. The source term is defined as:

\[
C_i = C_{i,0} e^{-\lambda t}
\]

- \(C_i\) is the concentration of the ith constituent at time \(t\) [M/L\(^3\)]
- \(C_{i,0}\) is the concentration of the ith constituent at the beginning of the stress period [M/L\(^3\)]
- \(\lambda\) is the first-order source decay rate constant [1/T]
- \(t\) is elapsed time since the beginning of the stress period [T]

The **Absolute Tolerance (ATOL)** and **Relative Tolerance (RTOL)** species parameters will be added for each species. These tolerance parameters are used by the differential equations solver in RT3D to control convergence errors while solving the applicable reaction model. Setting ATOL\((i) = 0.0\) results in a pure relative error test on Species \(i\), while setting
RTOL(i) = 0.0 results in a pure absolute error test on Species i. For practical problems, the following rules of thumb may be used to set ATOL and RTOL values:

- set RTOL(i) = 10^{-(m+1)}, where m is the desired number of significant digits for concentration output for species i ($C_i$)
- set ATOL(i) to a small value at which the absolute value of $C_i$ is essentially insignificant (typically between $10^{-6}$ to $10^{-9}$)

Please Note: that RTOL > ATOL > 0

Warning! Actual (global) errors may exceed the local tolerances, so choose ATOL(i) and RTOL(i) conservatively.

Variable Density Flow

The list of parameters required for a variable density flow model are presented below, associated base units are provided in square brackets [ ], where $L$ signifies length (e.g. meters, feet), $M$ signifies mass (e.g. kilograms, pounds), $T$ signifies time (e.g. seconds, days, years), and $\Theta$ signifies temperature (currently only degrees Celsius [oC] are supported, as these units are hard-coded into the governing equations implemented in SEAWAT). For more detailed information, see discussion on the variable density and viscosity packages.

- **Temperature**: this is a logical flag to select the species representing temperature. At most one species may be selected. This will link the selected temperature species to the viscosity package and change the concentration units to temperature. [-]
- **Initial Concentration**: same as above [see units item below]
- **CREF**: represents the reference concentration for the density and viscosity relationships for each species. [see units item below]
- **Units**: shows the project concentration units for all species [ML^3, except for the selected temperature species (if any) which will be shown in oC].
- **DRHODC ($\delta \rho / \delta C$)**: the density versus concentration slope [-], for the temperature species, it represents $\delta \rho / \delta T$
- **DMUxDC ($\delta \mu / \delta C$)**: the viscosity versus concentration slope [-], for the temperature species, it represents $\delta \mu / \delta T$, if the linear-relationship option is selected.
- **MDCOEFF**: species-specific diffusion coefficient [L^2/T]. **Note**: this is only used if the multi-diffusion option is set to true in the DSP package advanced settings.
Please Note: If you are developing a multi-species variable-density model, the major density-controlling species (typically salt) should be Species 1. Furthermore, if you are also modeling temperature, temperature should be species number 2.

10.1.3 Model Parameters

Model parameters consist of material properties relevant to fate and transport modeling associated with the geologic units that are independent of the constituent species to be modeled. The specific model parameters required as part of each simulation are defined by the retardation model and reaction model.

General Model Parameters:

IRCTOP
IRCTOP is an integer flag indicating how reaction variables are entered:
- By Cell (IRCTOP=2) [Default] - all reaction variables are specified as 3-D arrays on a cell-by-cell basis.
- By Layer (IRCTOP=0) - all reaction variables are specified as a 1-D array with each value in the array corresponding to a single layer. This option is mainly for retaining compatibility with the previous versions of MT3D.

IGETSC
IGETSC is an integer flag indicating whether the initial concentration for the non-equilibrium sorbed or immobile phase of all species should be read when non-equilibrium sorption (ISOTHM = 4) or dual-domain mass transfer (ISOTHM = 5 or 6) is simulated. The flag is set by answering the question:

Should sorbed or immobile-phase concentrations be read?

- No - then IGETSC = 0 and the initial concentration for the sorbed or immobile phase is not read. By default, the sorbed phase is assumed to be in equilibrium with the dissolved phase (ISOTHM = 4), and the immobile domain is assumed to have zero concentration (ISOTHM = 5 or 6).
- Yes - then IGETSC = 1 and the initial concentration for the sorbed phase or immobile liquid phase of all species will be read, as applicable.

Retardation Model - Model Parameters

Bulk Density
Soil Bulk Density is used to calculate the retardation coefficient for each chemical species according to the following formula:
\[ R_i = 1 + \frac{\rho_b}{n} \times K_{d,i} \]

where:

- \( R_i \) = Retardation Coefficient of Species \( i \) [\(-\)]
- \( \rho_b \) = Soil Bulk Density [M/L^3]
- \( n \) = Effective Soil Porosity [L^3/L^3] \( \rightarrow [\cdot] \)
- \( K_{d,i} \) = Distribution Coefficient of Species \( i \) [L^3/M]

The retardation coefficient is used to calculate the 'retarded' flow velocity (\( V_{R(i)} \)) of each chemical species according to the following formula:

\[ V_{x,i} = \frac{V}{R_i} \]

where:

- \( V_{R(i)} \) = Retarded Flow Velocity of Species \( i \) [L/T]
- \( V \) = Average Linear Groundwater Flow Velocity [L/T]
- \( R_i \) = Retardation Coefficient of Species \( i \) [\(-\)]

The retarded flow velocity is used to calculate the advective transport of each species.

Unless otherwise specified during the setup of the transport model, the default soil Bulk Density value for any new model created is 1700 kg/m^3.

If no sorption method is selected in the current transport variant, then no Bulk Density values are required for the simulation, and all of the options in the left-hand toolbar will be disabled.

**Reaction Model/Modules - Model Parameters**

The reaction modules associated with RT3D may require model parameter values and settings to be selected at the Define Modeling Objectives workflow step, these include:

- **Instantaneous aerobic degradation of BTEX** [IREACT=1];
- **Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors** [IREACT=3];
- **Rate-Limited Sorption** [IREACT=4];
- **Double-Monod Model** [IREACT=5];
- **Sequential First-Order Decay** [IREACT=6]; and
- **Aerobic/Anaerobic PCE/TCE Dechlorination** [IREACT=7]
where IREACT is a flag in the RT3D RCT package file whose value determines which reaction module is active in the transport model run.

**Please Note**: yield values associated with several reaction modules in RT3D are defined on a mg/L basis. To be consistent the user must use mg/L units for all concentrations when using these reaction modules. See additional notes for each reaction module.

### Instantaneous aerobic degradation of BTEX

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>Y_O2_BT_EX</td>
<td>Stoichiometric ratio of oxygen (consumed) to BTEX</td>
<td>3.14</td>
<td>[-]</td>
<td>Const</td>
<td>n/a Stoichiometry</td>
</tr>
</tbody>
</table>

*Note*: By changing the value of RC1 instantaneous reactions between any other chemicals can be simulated using this module

### Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>Cmax_Fe_2+</td>
<td>Maximum concentration of Fe2+</td>
<td>0</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td>R3-3(Reduction Capacities)</td>
</tr>
<tr>
<td>RC2</td>
<td>Cmax_CH_4</td>
<td>Maximum concentration of CH4</td>
<td>0</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td>R3-3(Rates)</td>
</tr>
<tr>
<td>RC3</td>
<td>K_O2</td>
<td>Hydrocarbon decay rate via aerobic process</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-3(Rates)</td>
</tr>
<tr>
<td>RC4</td>
<td>K_NO3</td>
<td>Hydrocarbon decay rate via denitrification</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-3(Rates)</td>
</tr>
<tr>
<td>RC5</td>
<td>K_Fe3+</td>
<td>Hydrocarbon decay rate via iron reduction</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-3(Rates)</td>
</tr>
<tr>
<td>RC6</td>
<td>K_SO4</td>
<td>Hydrocarbon decay rate via sulfate reduction</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-3(Saturation)</td>
</tr>
<tr>
<td>RC7</td>
<td>K_CH4</td>
<td>Hydrocarbon decay rate via methanogenesis</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-3(Saturation)</td>
</tr>
<tr>
<td>RC8</td>
<td>Ks_O2</td>
<td>Half-saturation constant for oxygen</td>
<td>0.5</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td>R3-3(Saturation)</td>
</tr>
</tbody>
</table>
### Parameter Description

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC9</td>
<td>Ks_NO3</td>
<td>Half-saturation constant for nitrate</td>
<td>0.5</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td>s)</td>
</tr>
<tr>
<td>RC10</td>
<td>Ks_Fe3+</td>
<td>Half-saturation constant for Fe3+</td>
<td>0.5</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC11</td>
<td>Ks_SO4</td>
<td>Half-saturation constant for sulfate</td>
<td>0.5</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC12</td>
<td>Ks_CH4</td>
<td>Half-saturation constant for methane</td>
<td>0.5</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC13</td>
<td>Ki_O2</td>
<td>Inhibition coefficient for the oxygen reaction</td>
<td>0.01</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC14</td>
<td>Ki_NO3</td>
<td>Inhibition coefficient for the nitrate reaction</td>
<td>0.01</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC15</td>
<td>Ki_Fe3+</td>
<td>Inhibition coefficient for the Fe3+ reaction</td>
<td>10</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC16</td>
<td>Ki_SO4</td>
<td>Inhibition coefficient for the sulfate reaction</td>
<td>0.01</td>
<td>[ML]^{-3}</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC17</td>
<td>Y_O2/BTEX</td>
<td>Stoichiometric ratio of oxygen (consumed) to BTEX</td>
<td>3.14</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
<tr>
<td>RC18</td>
<td>Y_NO3/BTEX</td>
<td>Stoichiometric ratio of nitrate (consumed) to BTEX</td>
<td>4.9</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
<tr>
<td>RC19</td>
<td>Y_Fe/BTEX</td>
<td>Stoichiometric ratio of Fe2+ (produced) to BTEX</td>
<td>21.8</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
<tr>
<td>RC20</td>
<td>Y_SO4/BTEX</td>
<td>Stoichiometric ratio of sulfate (consumed) to BTEX</td>
<td>4.7</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
<tr>
<td>RC21</td>
<td>Y_CH4/BTEX</td>
<td>Stoichiometric ratio of methane (produced) to BTEX</td>
<td>0.78</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
</tbody>
</table>

*Note: Yield values are on a mg/L basis; to be consistent, the user must use mg/L units for all concentrations when using this reaction module.

### Rate-Limited Sorption

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>K_mt</td>
<td>Mass-transfer rate coefficient</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-4(Sorption)</td>
</tr>
<tr>
<td>RC2</td>
<td>Kd</td>
<td>Linear partitioning coefficient (Kd)</td>
<td>1</td>
<td>[L^3/M]</td>
<td>Const/Var</td>
<td></td>
</tr>
</tbody>
</table>

### Double-Monod Model
### Numerical Modeling Workflow - Finite Difference Grids

**Parameter** | **Caption** | **Description** | **Default** | **Units** | **Constant / Spatially Variable** | **Node in Model Explorer**
--- | --- | --- | --- | --- | --- | ---
RC1 | \( \mu_m(w) \) | Specific utilization rate | 0 | \([1/T]\) | Const/Var | R3-5(Rates)
RC2 | \( K_s_d \) | Monod half-saturation constant for electron donor | 0.5 | \([\text{M/L}^3]\) | Const/Var | R3-5(Saturation s)
RC3 | \( K_s_a \) | Monod half-saturation constant for electron acceptor | 0.5 | \([\text{M/L}^3]\) | Const/Var |
RC4 | \( Y_{x/d} \) | Biomass produced per unit of electron donor utilized | 0 | [-] | Const | n/a Stoichiometry
RC5 | \( Y_{a/d} \) | Electron acceptor used per unit of electron donor utilized | 0 | [-] | Const | n/a Stoichiometry
RC6 | \( K_{\text{decay}} \) | First-order bacterial death or decay rate | 0 | \([1/T]\) | Const/Var | R3-5(Rates)
RC7 | \( K_{\text{attach}} \) | First-order bacterial attachment rate | 0 | \([1/T]\) | Const/Var |
RC8 | \( K_{\text{detach}} \) | First-order bacterial detachment rate | 0 | \([1/T]\) | Const/Var |

*Note*: This reaction module describes a general double Monod model. By setting appropriate yield and kinetic rate constants, users can model many types of biological systems. Kinetic constants for an aerobic system are given in Clement et al. (1998), and for an anaerobic denitrifying system are given in Clement et al. (1997). Also see Taylor and Jaffe (1990); Hornberger et al. (1992); Zysset et al. (1994); and Reddy and Ford (1996).

---

### Sequential First-Order Decay

**Parameter** | **Caption** | **Description** | **Default** | **Units** | **Constant / Spatially Variable** | **Node in Model Explorer**
--- | --- | --- | --- | --- | --- | ---
RC1 | \( K_A \) | PCE first-order degradation rate | 0 | \([1/T]\) | Const/Var | R3-6(Rates)
RC2 | \( K_B \) | TCE first-order degradation rate | 0 | \([1/T]\) | Const/Var |
RC3 | \( K_C \) | DCE first-order degradation rate | 0 | \([1/T]\) | Const/Var |
RC4 | \( K_D \) | VC first-order degradation rate | 0 | \([1/T]\) | Const/Var |
RC5 | \( Y_{\text{tce/pc}} \) | Yield coefficient, TCE/PCE | 0.792 | [-]’ | Const | n/a Stoichiometry
RC6 | \( Y_{\text{dce/tc}} \) | Yield coefficient, DCE/TCE | 0.738 | [-]’ | Const |
RC7 | \( Y_{\text{vc/dce}} \) | Yield coefficient, VC/DCE | 0.644 | [-]’ | Const |

*Note*: Yield values are on a mg/L basis; to be consistent, the user must use mg/L units for all concentrations when using this reaction module.
Aerobic/Aerobic PCE/TCE Dechlorination

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>Kp</td>
<td>Sequential (anaerobic) reaction rate for PCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-7(Rates)</td>
</tr>
<tr>
<td>RC2</td>
<td>Kt1</td>
<td>Sequential (anaerobic) reaction rate for TCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC3</td>
<td>Kt2</td>
<td>Aerobic decay rate for TCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC4</td>
<td>Kd1</td>
<td>Sequential (anaerobic) reaction rate for DCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC5</td>
<td>Kd2</td>
<td>Aerobic decay rate for DCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC6</td>
<td>Kv1</td>
<td>Sequential (anaerobic) reaction rate for VC</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC7</td>
<td>Kv2</td>
<td>Aerobic decay rate for VC</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC8</td>
<td>Ke1</td>
<td>Sequential (anaerobic) reaction rate for ETH</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC9</td>
<td>Ke2</td>
<td>Aerobic decay rate for ETH</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
</tbody>
</table>

*Note: All the yield values are fixed internally; to be consistent, use mg/L units for all species concentrations.

10.2 Define Numerical Model

The next step is to choose to Import Model (import Visual MODFLOW Classic project or MODFLOW files), or Create Grid (create an empty grid).
Import Model

You can import a Visual MODFLOW Classic project or a USGS MODFLOW 2000/2005 data set.

⚠️ Before You Start!

If you need to modify or maintain a model that utilizes any of the following features, you must continue to use Visual MODFLOW Classic interface for this:

- Flow Engines (MODFLOW-96)
- Transport Engines (MT3D99, PHT3D)
- Packages (ETS1, STR, MNW)
- MGO

When you click on the Import Model button, the following dialog will appear:
To import your Visual MODFLOW Classic project, select the .VMF file and click Open to continue.

To import a MODFLOW data set, change the file type to "MODFLOW 2000 or MODFLOW 2005", then select the desired .NAM file, and click Open to continue.

Once the model is finished importing, click \(\text{(Next Step)}\) to proceed.

Create Grid

If you select the Create Grid option, the following window will appear.
At this step, you can define the Horizontal Grid (grid size and the extents), and the Vertical Grid (number of layers and the layer elevations).

**Define Horizontal Grid**

Define the Grid size (number of Columns and Rows), Rotation, and the Grid Extents. If you prefer you can also define the Cell Height and Width.

---

**Maximum Grid Size**

There is no limit on the grid size you can use; however, if you design a grid that exceeds approximately 500,000 cells, it is strongly recommended that you use the 64-bit version, as more memory will be required to accommodate these model sizes. (With the 64-bit version of Visual MODFLOW Flex, running on 64-bit Windows and having at least 10 GB RAM, you should be able to create grids in excess of 5 million cells).
The grid extents can be defined manually; enter the min and max values for the X and Y coordinates.

Alternatively, the extents can be calculated from a bounding polygon. Select the option "Calculate extents from a polygon object". You will need to import a polygon object (from .SHP or .DXF File), that contains just a single, closed polygon, and provide this as an input. For details on importing polygon objects, see the section Import Polygons. Once the polygon is imported, it will appear in the Data tree in the top left.

Select/click on this polygon from the Data tree, then click on the blue arrow button under "Calculate extents from a polygon". The polygon should appear, and the X and Y extents will be calculated accordingly.

⚠️ Please Note: Visual MODFLOW Flex will calculate a minimum bounding rectangle from an irregularly-shaped polygon, in order to calculate the X and Y min/max extents.

**Define Vertical Grid**

Specify the number of layers you want for the grid. In the table below, you can manually define a constant (flat) layer elevation for the top of each layer, and the bottom of the bottommost layer (by default, Visual MODFLOW Flex will calculate equally thick layers). At this step, you can also defined varying layer elevations by using surfaces. This is intended for "simple" numerical models where "pancake-layer geology" is applicable. If your model has layers that pinchouts and/or discontinuities, it is advised that you follow the Conceptual Model workflow, as this will allow you to better accommodate more complex geology, using Horizon rules.

To define Surfaces for Grid layers, first you need to import surfaces (from Surfer .GRD, ESRI .ASC, etc.) or create surfaces from a set of points. Once the surfaces are available in the Data tree, select each surface for the appropriate layer elevation, being sure to work your way downwards (define top of layer 1 first, then work downwards).
Using Surfaces

When using surfaces to define grid layer elevations, please keep in mind the following restrictions:

- All surfaces must be large enough (in area) to cover completely the grid extents
- Surfaces must not intersect with other surfaces (e.g. above or below). Fix these surfaces, or use the Conceptual Modeling workflow to accommodate these layers.
- Surfaces must be added in the correct order (vertically), working from top downwards.

Once you are finished defining the horizontal and vertical grid, click on the [Create Grid] button at the top (middle) of the window.

Once the grid is created, there are several options to customize the grid to your project needs.

The Edit Grid button will load a window where you can refine/coarsen the grid. For more details on this option, please see the Edit Grid section.

The Define Child Grid button will load a window where you can create child (local) grids for a MODFLOW-LGR run. For more details, please see the Define Child Grid (for LGR) section.

When you are finished with the grid edits, click on the (Next Step) to proceed. This will generate the model tree structure (in the lower left corner of the window), with Inputs and Outputs.

10.3 View/Edit Grid

After you have created a grid or imported a grid (and associated model), you will be taken to the View/Edit Grid step. There are several options available.

- Under Views, select the various views you want to see in the Flex viewer; Visual MODFLOW Flex allows you to simultaneously show a layer, row, column and 3D Views.
- Adjust a specific layer, row, or column using the up/down arrows or enter a specific row, column, or layer integer. Alternatively, click on the button then click on any specific row, column, or layer in any of the 2D views, and the selected row, column, or layer will be set automatically.
- The standard navigation tools allow you to zoom, pan, and in the case of 3D view, rotate.

You can right-click on the numerical grid in the Model Explorer and select Settings. In addition, the Numerical Grid can be added to a stand-alone 3D view; to do this, right click and select 3D Viewer.
There are several other options available on the right click menu.

The numerical grid can be exported to shapefile; see Export for more details.

- **Export Layers**: allows you to export the layers of the grid to .txt or .csv file. These files can be edited and then imported back.
- **Import Layers**: allows you to import back the layers you exported (and edited).

You will also find the option to "Make Layers Surfaces" which will create a folder in the Data Explorer (named: grid name Layers Surfaces) where you will find that surfaces have been created for each layer starting with Top, followed by Bottom.001, Bottom.002 etc.

**Grid Modification**

In the toolbox under the Grid Refinement section you have options to Edit Numerical Grid... (horizontal) Edit Grid Layers... (vertical) as well as Define Child Grid (for LGR) and Extract subgrid.

**Assign Inactive Cells**

At this step, you can digitize a polyline or polygon, and set the selected cells as inactive (or conversely, change inactive cells back to active cells)

- Select the desired flag from the combo box under "Inactive Cells" (make Active cells inactive, or make Inactive cells Active)
- Select the Assign button.
- Choose the desired shape for drawing (Single, Polyline, or Polygon)

Use the left mouse button to start drawing in the grid view, and add highlight the desired cells; for single cell selection, a point will appear on top of the selected cell.
- Once you have finished digitizing, right click with the mouse and select "Define Attributes" (or click on the "Finish" button under the toolbox); the following window will appear.
Choose which layers you wish to include in the cell assignments; by default, just the current layer is selected, however you can select other (or all) layers from this window.

- Click OK
- The selected cells will be set as Inactive or Active, depending on the option you have selected. Inactive cells will be colored turquoise.

When you are finished with viewing the grid, click (Next Step) to proceed to the Define Properties step.

10.4 Define Properties

At this step, you can assign, edit and copy the flow properties for the model.
You can assign properties while viewing either by Layer, Row, or Column.

Under the Toolbox, use the combo box to select from the available property groups:

- **Flow Parameters**
  - **Conductivity**: includes horizontal conductivity in the X direction (Kxx), in the Y-direction (Kyy), and vertical conductivity (Kzz)
  - **Storage**: includes Specific Storage (Ss), Specific Yield (Sy), Total Porosity (Tp), and Effective Porosity (Ep)
  - **Initial Heads**
  - **Vadose Zone**: only available/required if using MODFLOW-SURFACT or MODFLOW-2005/MODFLOW-NWT with the UZF package

When transport is included with your model, the following additional property groups may be included:

- **Transport Parameters**
  - **Initial Concentration** Group - one node for each Species
  - **Species Parameters** Group(s)* - one or more node for each Species
  - **Model Parameters** Group(s)*
Longitudinal Dispersion

*Please Note*: the number of Species Parameters and Model Parameters (if any) will depend on what Sorption and Reaction options you have defined in the Define Modeling Objectives.

For each parameter group, you can choose to render by Zones or by a selected attribute. Based on your selection, the color rendering in the views will change.

Use the same tools as described in the previous step to manipulate the views.

The display tools will allow you to switch from discrete cells rendering to color shading/contours (note, this is available only when you do attribute rendering, and not when you are rendering by ZoneID).

At the bottom of the display, you will see in the status bar the position of your mouse cursor in the current view (XY) grid position (Layer, Row, Column), and the Zone ID or attribute value for the selected cell.

For more details on the required flow and transport parameters for each category, please see section: Background on Flow and Transport Properties

Assign

Property zone values can be assigned directly to selected numerical cells using the Assign options from the toolbox.

- **Single**: Select one or more cells with the left mouse button; a small red point will appear over the selected cell
- **Polyline**: Draw one or more polylines to select the desired cells
- **Polygon**: Draw one or more polygons to select the desired cells
- **Entire Layer/Row/Column**: Selects the currently selected layer/row/column (all active cells)
- **Using Data Object**: Use data object (polygon or polyline) to select the desired cells

When you use either the Assign/Polygon or Assign/Entire Layer option, the property zone will be assigned to the numerical grid and a corresponding conceptual property zone representation will be created; this is useful when you re-run the Conceptual to Numerical Conversion option for the current (or new) numerical grid.

However, when you use the Assign/Single or Assign/Polyline options, no conceptual property zone is created; please be aware of this limitation should you choose to create a new
numerical model from the conceptual model; the property values in this new model run may need to be adjusted.

After you have done the desired cell selections, right click with the mouse to close the shape and select "Finish".

The New Property Zone window will appear as shown below.

In the top left, select which zone the selected cells should be assigned to (from the combo box). If you choose an existing zone, the parameter values cannot be adjusted from within this GUI (instead, select the Edit option described above, to adjust parameter values for existing zones.)

Or, click the [New] button to create a new property zone. After doing this, a new zone number will be incremented. Next, select in which layers the new property zone should be generated, by default, your current layer will be selected, however you can include other layers or select all layers.

Lastly, define the "Zone Attributes" for this new property zone in the main grid. There are several "Methods" for defining the attributes, and these include Constant Value and Use Surface (Use 3D Gridded Data and Use Shape Data are not applicable for this scenario).

**Constant Value**
The Constant Value method is selected by default for each parameter in the data input grid and allows you to specify a spatially constant value for the parameter. Replace the default value with a new value.

**Use Surface**
The Use Surface method allows you use an existing surface data object to define spatially-variable attribute values. This is ideal if you have a Surfer .GRD or ESRI ASCII GRD file containing parameter values. Follow the steps below:

- Select "Surface" as the method from the combo box. You should then see the button become beside the "Value" column
- From the Data Explorer, select the desired surface data object
- Click the button to insert this data object into the Object field.

Once you are finished, click OK and the new property zone will be created. (note, when you view the Database values for this new property zone, it will show "Distributed Values" since it contains a range of values for this zone, not just a single value).

Use 3D Gridded Data
You may want to define initial concentrations from a .UCN file from a previous transport run. In order to use this option, you must need to import the .UCN file to the project, as a "3D Gridded Data" set. Follow these steps in the section "Import 3D Gridded Data". Once the file is imported, you select this object when defining the properties.

- Use assign polygon or entire layer, to select the desired group of cells
- Click the [New] button to create a new zone.
- Select "Use 3D Gridded Data" as the method from the combo box.
- You should then see the button become activated beside the "Value" column
- From the Data Explorer, select the desired 3D Gridded Data (which contains the imported .UCN file)
- Click the button to insert this data object into the Object field.
- By default, Concentration should be selected in the "Mapping" column (you may need to re-select this from the combo box).
- Once you are finished, click OK and the new property zone will be created.
- You can view the Initial Concentration rendering for the full 3D grid, by changing the rendering type from "Zone" to "Initial Concentration"

Please Note: when you view the Database values for this new property zone, it will show "Distributed Values" since it contains a range of values for this zone, not just a single value)

Edit
Select the Edit button to edit Property zone values after they have been assigned and the Edit property dialog is launched. Here you can select the zone and then you will have the ability to edit any of the cells that are part of the zone.
The following buttons are available to assist with your edits:

- Assign to current cell
- Assign to column
- Apply to selected cells

Also available is the Script tab. If preferred you can make your edits by using the Script option. On this tab you will find the script for any edits you make on the Edit Cells tab allowing you to learn the script language. You can then use the script to "replay" the same edits again.

You can also build expressions using the Expression Builder.

**Copy**

Select the Copy button to copy property values from the current Layer, Row, or Column based on the active view (i.e. the active view is highlighted in blue) to other layer(s), row(s), or column(s). Note that you can click in a view to make it the active view. Also note that the currently active layer, row, or column will be filtered from the list.
Adjusting Style Settings

Each group of flow properties exists as a separate item in the model tree, as shown below.

Right click on each node, and select Settings. This will allow you to modify the appearance of each group of flow properties.
In addition, each property group can be added to a stand-alone 3D view; to do this, right click and select 3D-Viewer.

The 2D Colors and Contour line settings can be adjusted. For more details, see Color and Contour settings.

**Export Properties**

Any of the model properties can be exported to shapefile; see Export for more details.

### 10.4.1 Theory

The following sections present an overview of the property zone parameters required for flow and transport models in Visual MODFLOW Flex.

A steady-state groundwater flow model requires conductivity and initial heads property values for each active grid cell in order to run a flow simulation, while transient flow models also require storage properties for each active grid cell. Similarly, a transport model requires various model and species parameter values for each active grid cell in order to run a transport simulation. Upon creating a Visual MODFLOW Flex project, the default flow and transport parameter values are assigned to every grid cell in the model domain. This will ensure the model has the minimum data required to run a simulation. However, in most situations, the flow and transport properties will not be uniform throughout the entire model domain, and it will be necessary to assign different property values to different areas of the model.
Heterogeneous property values are supported by Visual MODFLOW Flex using either Constant Value Property Zones, or Distributed Value Property Zones. These two different approaches are described below.

**Constant Value Property Zones**

The Constant Value Property Zones approach is the most simple and straight-forward, and can be used for all model properties supported by Visual MODFLOW Flex. Different model properties are accommodated by grouping grid cells sharing the same property values into “property zones”. Each property zone will (normally) contain a unique set of property values, and is represented by a different grid cell color.

The Constant Value Property Zones approach requires the development of a conceptual model, whereby each hydrostratigraphic unit of the model is assigned a uniform set of property values. For example, consider an aquifer where there is pumping test data and slug test data indicating a range of horizontal conductivity values from $1 \times 10^{-4}$ cm/s to $5 \times 10^{-4}$ cm/s at different locations within the aquifer. The conceptual approach would assign a uniform $K_x$ and $K_y$ value of $2.5 \times 10^{-4}$ cm/s to the entire aquifer. This value would be adjusted up or down for calibration purposes within the range of values reported. If a reasonable calibration cannot be achieved using this conceptual model, it may be necessary to sub-divide this region into several zones to accommodate local irregularities in the flow pattern. However, almost all modeling textbooks strongly recommend starting out simple at first, getting as close a solution as possible, and then making the model more complex, if necessary.

**Distributed Value Property Zones**

The Distributed Value Property Zones approach is available for flow properties (conductivity, storage, and initial Heads), and most transport properties (e.g. initial concentrations, longitudinal dispersivity, sorption parameters). This approach is a little more complicated than the constant value property zone approach because it involves linking a property zone to one or more parameter distribution arrays containing data interpolated from scattered observation points. When a property zone is linked to a distribution array, the property values assigned to each grid cell within that zone are calculated by multiplying the zone parameter value with the corresponding value from the parameter distribution array. If the grid spacing from the model does not match the grid spacing from the distribution array, a bivariate interpolation scheme is used to calculate the appropriate parameter value at the center of the model grid cell using the four nearest data nodes in the parameter distribution array.

**Flow Properties**

**Conductivity**

- $K_x$ - Hydraulic conductivity in the direction of the model X-axis
- $K_y$ - Hydraulic conductivity in the direction of the model Y-axis
- $K_z$ - Hydraulic conductivity in the direction of the model Z-axis
These conductivity parameters may be defined on a cell-by-cell basis using constant property values and/or distributed property values. When importing or assigning the conductivity property zones, Visual MODFLOW Flex will require valid data for each of the above parameters.

**Anisotropy**

The reason Visual MODFLOW Flex prompts for both Kx and Ky is because there are two options for defining the horizontal anisotropy of the Conductivity property zones:

- Anisotropy by layer
- Anisotropy as specified

![Please Note:](image-url) The anisotropy option is set in Translation settings. (see Anisotropy for more details).

If the 'Anisotropy by layer' option is used, the Kx value will determine the conductivity in the X-direction, and the specified anisotropy ratio (Ky/Kx) for each layer will be used to calculate the Ky value for each grid cell.

If the 'Anisotropy as specified' option is used, the model will use the Kx and Ky values defined for each property zone.

**Storage**

- Ss - specific storage
- Sy - specific yield
- Eff. Por - effective porosity
- Tot. Por - total porosity

**Specific Storage (Ss)** is defined as the volume of water that a unit volume of aquifer releases from storage under a unit decline in hydraulic head due to aquifer compaction and water expansion. Visual MODFLOW Flex determines the primary storage coefficient (sf1) for MODFLOW based on the user defined Specific Storage parameter. The primary storage coefficient is calculated by Visual MODFLOW to be equal to the specific storage multiplied by the layer thickness (Specific Storage x thickness = Primary storage coefficient). Specific Storage is is only used in transient simulations.

**Specific Yield (Sy)** is known as the storage term for an unconfined aquifer. It is defined as the volume of water that an unconfined aquifer releases from storage per unit surface area per unit decline in the water table. For sand and gravel aquifers, specific yield is generally equal to the porosity. MODFLOW uses Ss or Sy depending on the layer type assigned by the user (please refer to "Layer Type Settings"). For an unconfined layer, MODFLOW uses Sy to determine storage volumes. For a confined layer, Ss is used. For a variable layer, MODFLOW will check the head value of the cell to determine if it is confined or not. If you do not have measured parameter values for Sc and Sy it is recommended that you refer to literature values as a default. Specific Yield is only used in transient simulations.

**Effective Porosity (Eff. Por)** is the pore space through which flow actually occurs, and is used by the particle tracking algorithms (MODPATH/MOD-PATH3DU) to determine the
average linear groundwater velocities for use in time-dependent capture zones and time markers along pathlines. This term is not used for MODFLOW simulations.

**Total Porosity (Tot. Por)** is the percentage of the rock or soil that is void of material, and is used by MT3D/RT3D/SEAWAT (by default) to determine the chemical reaction coefficients, and for calculating the average linear groundwater flow velocity in the particle tracking solution schemes. A different porosity is used for MT3D/RT3D/SEAWAT than for particle tracking because the transport engines account for additional transport and reactive processes, such as dispersion. The total porosity term is not used for MODFLOW simulations.

These storage parameters may be defined on a cell-by-cell basis using constant property values and/or distributed property values. When importing or assigning the storage property zones, Visual MODFLOW Flex will require valid data for each of the above parameters.

ℹ️ **Please Note**: MODFLOW-SURFACT supports the time-varying material properties (TMP1) package (as an add-on) where the flow properties may change over time, as discussed [here](#).

**Initial Heads**

In order to start solving the flow simulation, flow engines require an initial “guess” for the head values in the model. A good initial guess for the starting heads of the simulation can reduce the required run time significantly, while a poor initial guess can increase run times or lead to stability and/or convergence issues. The Initial Head values are also used to calculate drawdown values, as measured by the difference between the starting head and the calculated head.

**Vadose Zone**

The vadose zone object is only available when an unsaturated flow model (MODFLOW-2005 with the unsaturated zone flow (UZF) package or MODFLOW-SURFACT) is selected in the **Modeling Objectives** workflow step.

The vadose zone, or unsaturated zone, extends from the land surface to the water table. The vadose zone typically has a lower hydraulic conductivity because some of the pore space is filled with air, and the soil moisture in the vadose zone only travels through the wetted cross-section of the pore space. The relative proportion of air to water in the pores can vary, and consequently the hydraulic properties of the porous media can also vary.

In general, soil moisture in the vadose zone is under tension, and it travels through the vadose zone due to total soil-moisture potential (the sum of the elevation potential, and the pressure head). The pressure head is a function of the volumetric water content of the soil, and depends upon whether the soil has previously undergone wetting or drying. The relationship between pressure head and volumetric water content for a particular soil is known as a soil-water characteristic curve. Using the experimental soil-water characteristic curve and the soil hydrologic parameters, the movement of water in the vadose zone can be determined using the BCF4 package in MODFLOW-SURFACT or the UZF package in MODFLOW 2005. A sample index of soil properties is given below:
**Table: Sample Index of Soil Hydraulic Properties by Soil Texture**

<table>
<thead>
<tr>
<th>Soil Type</th>
<th>Sand [%]</th>
<th>Clay [%]</th>
<th>Bulk Density [g/cm³]</th>
<th>Field Capacity [cm³/cm³]</th>
<th>Wilting Point [cm³/cm³]</th>
<th>Porosity Fraction [-]</th>
<th>Saturated Hydraulic Conductivity [cm/s]</th>
<th>Slope of Retention Curve [log space]**</th>
<th>Alpha parameter [cm⁻¹] ***</th>
</tr>
</thead>
<tbody>
<tr>
<td>sand (s)</td>
<td>94.83</td>
<td>2.27</td>
<td>1.49</td>
<td>0.08</td>
<td>0.03</td>
<td>0.43</td>
<td>0.0107</td>
<td>4.1</td>
<td>0.145</td>
</tr>
<tr>
<td>loamy sand (ls)</td>
<td>85.23</td>
<td>6.53</td>
<td>1.52</td>
<td>0.15</td>
<td>0.06</td>
<td>0.42</td>
<td>0.003</td>
<td>3.99</td>
<td>0.124</td>
</tr>
<tr>
<td>sandy loam (sl)</td>
<td>69.28</td>
<td>12.48</td>
<td>1.57</td>
<td>0.21</td>
<td>0.09</td>
<td>0.4</td>
<td>0.0015</td>
<td>4.84</td>
<td>0.075</td>
</tr>
<tr>
<td>silt loam (sil)</td>
<td>19.28</td>
<td>17.11</td>
<td>1.42</td>
<td>0.32</td>
<td>0.12</td>
<td>0.46</td>
<td>0.0011</td>
<td>3.79</td>
<td>0.02</td>
</tr>
<tr>
<td>silt (si)</td>
<td>4.5</td>
<td>8.3</td>
<td>1.28</td>
<td>0.28</td>
<td>0.08</td>
<td>0.52</td>
<td>0.0023</td>
<td>3.05</td>
<td>0.016</td>
</tr>
<tr>
<td>loam (l)</td>
<td>41</td>
<td>20.69</td>
<td>1.49</td>
<td>0.29</td>
<td>0.14</td>
<td>0.43</td>
<td>0.0005</td>
<td>5.3</td>
<td>0.036</td>
</tr>
<tr>
<td>sandy clay loam (scl)</td>
<td>60.97</td>
<td>26.33</td>
<td>1.6</td>
<td>0.27</td>
<td>0.17</td>
<td>0.39</td>
<td>0.0007</td>
<td>8.66</td>
<td>0.059</td>
</tr>
<tr>
<td>silty clay loam (sicl)</td>
<td>9.04</td>
<td>33.05</td>
<td>1.38</td>
<td>0.36</td>
<td>0.21</td>
<td>0.48</td>
<td>0.0015</td>
<td>7.48</td>
<td>0.01</td>
</tr>
<tr>
<td>clay loam (cl)</td>
<td>30.08</td>
<td>33.46</td>
<td>1.43</td>
<td>0.34</td>
<td>0.21</td>
<td>0.46</td>
<td>0.0005</td>
<td>8.02</td>
<td>0.019</td>
</tr>
<tr>
<td>sandy clay (sc)</td>
<td>50.32</td>
<td>39.3</td>
<td>1.57</td>
<td>0.31</td>
<td>0.23</td>
<td>0.41</td>
<td>0.0003</td>
<td>13</td>
<td>0.027</td>
</tr>
<tr>
<td>silty clay (sic)</td>
<td>8.18</td>
<td>44.58</td>
<td>1.35</td>
<td>0.37</td>
<td>0.25</td>
<td>0.49</td>
<td>0.0008</td>
<td>9.76</td>
<td>0.005</td>
</tr>
<tr>
<td>clay (c)</td>
<td>24.71</td>
<td>52.46</td>
<td>1.39</td>
<td>0.36</td>
<td>0.27</td>
<td>0.47</td>
<td>0.0009</td>
<td>12.28</td>
<td></td>
</tr>
</tbody>
</table>

* Source: "Average hydraulic conductivity properties of ARS soil texture classes", draft dated February, 2000 by J. Schaake. This expanded the work of others and included a total of 2128 soil samples. Wilting point is the fractional water content at 15 bar tension; field capacity is the fractional water content at 1/3 bar tension.


** Average values of the van Genuchten soil parameters obtained by experimental means (Carsel and Parrish, 1988) and also reported on page 180 in Contaminant Hydrogeology book by C. W. Fetter (1999). You may want to consider also the ROSETTA code for estimating the unsaturated soil hydraulic properties which can be downloaded from the US Salinity Laboratory website: [http://www.ussl.ars.usda.gov](http://www.ussl.ars.usda.gov).
Unsaturated Flow - UZF Package

In order to solve the variably saturated flow problem using the UZF package, it is necessary to specify the relationship of relative permeability \( K_{rw} \) versus water phase saturation \( \theta \), and pressure head \( \psi \) versus water phase saturation \( \theta \). Flow through and storage in the unsaturated zone is estimated in the UZF package using a kinematic wave approximation of Richard's equation using the method of characteristics. The approach includes the assumptions that flow in the unsaturated zone is driven by gravity, that negative potential gradients are ignored, and that flow is vertical through a homogenous column of soil. The Brooks-Corey function is used to estimate the relationship between relative hydraulic conductivity and water saturation:

\[
K_{rw} = S_e^{\epsilon} = \left[ \frac{\theta - \theta_r}{1 - \theta_r} \right]^\epsilon
\]

where:

- \( K_{rw} \) is the relative hydraulic conductivity \([L/T]\)
- \( \epsilon \) is the Brooks-Corey exponent \([-]\)
- \( S_e \) is the effective water saturation \([-]\)
\( \theta \) is the current water saturation, which is a function of pressure head [\( \cdot \)]

\( \theta_r \) is the residual water saturation [\( \cdot \)]

The parameters are entered as follows:

**EPS** - (Brooks-Corey exponent \( \varepsilon \)) controls the air-entry pressure - the smaller the \( \alpha \) value, the larger the capillary fringe above the water table. Gravels and sands have a large \( \alpha \) while clayey soils have a small \( \alpha \).

**THTi** - (initial water saturation \( \theta_i \)) initial water content in the vadose zone.

**THTs** - (residual water saturation \( \theta_r \)) depends upon several factors of the soil including packing. Residual saturation values are generally low for sands and gravels and high for clayey soils. Residual saturation typically ranges from 0.01 to 0.4.

The BCF4 package in MODFLOW SURFACT can solve for variably saturated flow in the vadose zone in three dimensions using the Brooks-Corey equation described above or using the Van Genuchten (1980) equation for relative permeability:

\[
K_{rw} = S_o^{0.5} \left[ 1 - \left( 1 - S_o \frac{1}{\psi} \right)^{\gamma} \right]^2
\]

where:

\( \gamma \) is an empirical parameter [dimensionless] and is related to empirical parameter \( \beta \) as follows: \( \gamma = 1 - 1/\beta \)

The relationship of the pressure head (\( \psi \)) [L] to relative saturation is described by van Genuchten (1980) using the following equation:

\[
S_o = \begin{cases} \frac{\theta - \theta_r}{1 - \theta_r} & \text{for } \psi < 0 \\ \frac{1}{1 - \theta_r} & \text{for } \psi \geq 0 \end{cases}
\]

where:

\( \alpha \) is an empirical parameter [1/L]

\( \beta \) is an empirical parameter [-]

\( h_c \) is the capillary pressure [L] and is related to pressure head (\( \psi \)) and elevation (\( z \)) by \( \psi = h_c + z \)
Depending on the saturation model selected, the primary inputs for simulating groundwater flow in the vadose zone using MODFLOW-SURFACT are the van Genuchten parameters (\(\alpha\), \(\beta\), and \(\theta_r\)) and the Brooks-Corey parameter (\(\varepsilon\)). The Brooks-Corey and van Genuchten empirical parameters can be measured in the laboratory for a given soil by plotting a soil-water characteristic curve.

The parameters are entered as follows:

- **VANAL** - (van Genuchten \(\alpha\) parameter) controls the air-entry pressure - the smaller the \(\alpha\) value, the larger the capillary fringe above the water table. Gravels and sands have a large \(\alpha\) while clayey soils have a small \(\alpha\).

- **VANBT** - (van Genuchten \(\beta\) parameter) controls how rapidly the saturation drops from unity to residual, and is an indicator of the grain size distribution. For larger values of \(\beta\), the saturation drops rapidly with head (uniform grain size distribution), while for smaller values of \(\beta\), the drop is more gradual (grain-size, and therefore pore size is more varied). \(\beta\) is greater than unity, but typically can be from 1.3 to 6, depending upon the soil.

- **VANSR** - (residual water saturation \(\theta_r\)) depends upon several factors of the soil including packing. Residual saturation values are generally low for sands and gravels and high for clayey soils. Residual saturation typically ranges from 0.01 to 0.4.

- **Brook** - (Brooks-Corey \(\varepsilon\)) is the Brooks-Corey empirical exponent. It must be greater than zero (default value is 0.5).

The BCF4 package in SURFACT can also be used to simulate vapor flow through the vadose zone. Pressure head in the air phase \((P_a)\) and total head in the air phase \(h_{ap}\) are related as follows:

\[
h_{ap} = \frac{P_a}{\rho_w g} = h_{ap} - \frac{P_a z}{\rho_w}
\]

where

- \(\rho_w\) is density of water \([L^3/T]\)
- \(\rho_a\) is the density of air \([L^3/T]\)
- \(g\) is the gravitational acceleration constant \([L/T^2]\)
- \(h_{ap}\) is the total head in the air phase \([L]\)
- \(z\) is the elevation \([L]\)

Soil vapor flow options for MODFLOW-SURFACT simulations are set using the advanced settings for the BCF4 package in the Translation Settings workflow step. These are initially based on the project defaults for physical constants in the Model Defaults tab (found under File / Project Defaults / Model Defaults) and include the following:
RHOWP ($\rho_w$) is density of water [L$^3$/T]

PHOAP ($\rho_a$) is the density of air [L$^3$/T]

VISW ($\mu_w$) is the viscosity of water [M/LT]

VISG ($\mu_a$) is the viscosity of air [M/LT]

COMPWAT ($\beta_w$) compressibility of water [LT$^2$/M]

COMPAIR ($\beta_a$) compressibility of air [LT$^2$/M]

ATMGP ($P_{atm}$) is the standard atmospheric pressure [M/LT$^2$]

GRAV (g) is the gravitational acceleration constant [L/T$^2$]

Transport Properties

If no transport engine variant has been selected, the Transport Properties options will be disabled. If a transport engine has been selected then the following properties must be defined.

Initial Concentration

In many cases, the historical conditions of the site are unknown, and the contaminant source has been removed or remediated. In such cases, the groundwater contamination is still present and the mass transport simulation must be run forward in time, starting from the existing conditions, to predict the potential downstream impacts. The Initial Concentration properties define the existing conditions of each chemical species at the start of the simulation period.

Initial Concentrations must be defined for each chemical species that you have defined in the Define Modeling Objectives; the default value is 0.

Model Parameters

Model parameters consist of material properties associated with the geologic units that are independent of the constituent species to be modeled. The specific model parameters required as part of each simulation is defined by the retardation model and reaction model selected as part of the Define Objectives workflow step.

Retardation Model - Model Parameters

Bulk Density

Soil Bulk Density is used to calculate the retardation coefficient for each chemical species according to the following formula:
\[ R_i = 1 + \frac{\rho_b}{n} \times K_{d(i)} \]

where:

- \( R_i \) = Retardation Coefficient of Species i [-]
- \( \rho_b \) = Soil Bulk Density [M/L^3]
- \( n \) = Effective Soil Porosity [L^3/L^3] \to [-]
- \( K_{d(i)} \) = Distribution Coefficient of Species i [L^3/M]

The retardation coefficient is used to calculate the 'retarded' flow velocity (\( V_{R(i)} \)) of each chemical species according to the following formula:

\[ V_{R(i)} = \frac{V}{R_i} \]

where:

- \( V_{R(i)} \) = Retarded Flow Velocity of Species i [L/T]
- \( V \) = Average Linear Groundwater Flow Velocity [L/T]
- \( R_i \) = Retardation Coefficient of Species i [-]

The retarded flow velocity is used to calculate the advective transport of each species.

Unless otherwise specified during the setup of the transport model, the default soil Bulk Density value for any new model created is 1700 kg/m^3.

If no sorption method is selected in the current transport variant, then no Bulk Density values are required for the simulation, and all of the options in the left-hand toolbar will be disabled.

**Reaction Model/Modules - Model Parameters**

The reaction modules associated with RT3D may require model parameter values and settings to be selected at the Define Modeling Objectives workflow step, these include:

- **Instantaneous aerobic degradation of BTEX** [IREACT=1];
- **Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors** [IREACT=3];
- **Rate-Limited Sorption** [IREACT=4];
- **Double-Monod Model** [IREACT=5];
- **Sequential First-Order Decay** [IREACT=6]; and
- **Aerobic/Anaerobic PCE/TCE Dechlorination** [IREACT=7]
where IREACT is a flag in the RT3D RCT package file whose value determines which reaction module is active in the transport model run.

**Please Note**: yield values associated with several reaction modules in RT3D are defined on a mg/L basis. To be consistent the user must use mg/L units for all concentrations when using these reaction modules. See additional notes for each reaction module.

*Instantaneous aerobic degradation of BTEX*

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>$Y_{\text{O}_2/\text{BT EX}}$</td>
<td>Stoichiometric ratio of oxygen (consumed) to BTEX</td>
<td>3.14</td>
<td>[-]</td>
<td>Const</td>
<td>n/a</td>
</tr>
</tbody>
</table>

*Note*: By changing the value of RC1 instantaneous reactions between any other chemicals can be simulated using this module.

*Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors*

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>$C_{\text{max}_\text{Fe}^{2+}}$</td>
<td>Maximum concentration of Fe$^{2+}$</td>
<td>0</td>
<td>[M/L$^3$]</td>
<td>Const/Var</td>
<td>R3-3(Reduction Capacities)</td>
</tr>
<tr>
<td>RC2</td>
<td>$C_{\text{max}_\text{CH}_4}$</td>
<td>Maximum concentration of CH$^4$</td>
<td>0</td>
<td>[M/L$^3$]</td>
<td>Const/Var</td>
<td>R3-3(Rates)</td>
</tr>
<tr>
<td>RC3</td>
<td>$K_{\text{O}_2}$</td>
<td>Hydrocarbon decay rate via aerobic process</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC4</td>
<td>$K_{\text{NO}_3}$</td>
<td>Hydrocarbon decay rate via denitrification</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC5</td>
<td>$K_{\text{Fe}^{3+}}$</td>
<td>Hydrocarbon decay rate via iron reduction</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC6</td>
<td>$K_{\text{SO}_4}$</td>
<td>Hydrocarbon decay rate via sulfate reduction</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC7</td>
<td>$K_{\text{CH}_4}$</td>
<td>Hydrocarbon decay rate via methanogenesis</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC8</td>
<td>$K_{s_{\text{O}_2}}$</td>
<td>Half-saturation constant for oxygen</td>
<td>0.5</td>
<td>[M/L$^3$]</td>
<td>Const/Var</td>
<td>R3-3(Saturation)</td>
</tr>
</tbody>
</table>
### Numerical Modeling Workflow - Finite Difference Grids

#### Parameter, Caption, Description, Default, Units, Constant / Spatially Variable

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC9</td>
<td>Ks_NO3</td>
<td>Half-saturation constant for nitrate</td>
<td>0.5</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td>R3-4(Nitrate)</td>
</tr>
<tr>
<td>RC10</td>
<td>Ks_Fe3+</td>
<td>Half-saturation constant for Fe3+</td>
<td>0.5</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC11</td>
<td>Ks_SO4</td>
<td>Half-saturation constant for sulfate</td>
<td>0.5</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC12</td>
<td>Ks_CH4</td>
<td>Half-saturation constant for methane</td>
<td>0.5</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC13</td>
<td>Ki_O2</td>
<td>Inhibition coefficient for the oxygen reaction</td>
<td>0.01</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC14</td>
<td>Ki_NO3</td>
<td>Inhibition coefficient for the nitrate reaction</td>
<td>0.01</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC15</td>
<td>Ki_Fe3+</td>
<td>Inhibition coefficient for the Fe3+ reaction</td>
<td>10</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC16</td>
<td>Ki_SO4</td>
<td>Inhibition coefficient for the sulfate reaction</td>
<td>0.01</td>
<td>[M/L^3]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC17</td>
<td>Y_O2/BTEX</td>
<td>Stoichiometric ratio of oxygen (consumed) to BTEX</td>
<td>3.14</td>
<td>[-]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC18</td>
<td>Y_NO3/BTEX</td>
<td>Stoichiometric ratio of nitrate (consumed) to BTEX</td>
<td>4.9</td>
<td>[-]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC19</td>
<td>Y_Fe/BTEX</td>
<td>Stoichiometric ratio of Fe2+ (produced) to BTEX</td>
<td>21.8</td>
<td>[-]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC20</td>
<td>Y_SO4/BTEX</td>
<td>Stoichiometric ratio of sulfate (consumed) to BTEX</td>
<td>4.7</td>
<td>[-]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC21</td>
<td>Y_CH4/BTEX</td>
<td>Stoichiometric ratio of methane (produced) to BTEX</td>
<td>0.78</td>
<td>[-]</td>
<td>Const/Var</td>
<td></td>
</tr>
</tbody>
</table>

*Note: Yield values are on a mg/L basis; to be consistent, the user must use mg/L units for all concentrations when using this reaction module.

#### Rate-Limited Sorption

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>K_mnt</td>
<td>Mass-transfer rate coefficient</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC2</td>
<td>Kd</td>
<td>Linear partitioning coefficient (Kd)</td>
<td>1</td>
<td>[L^3/M]</td>
<td>Const/Var</td>
<td></td>
</tr>
</tbody>
</table>

#### Double-Monod Model

© 2019 by Waterloo Hydrogeologic
### Biological Reactions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>$\mu_m(w)$</td>
<td>Specific utilization rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-5(Rates)</td>
</tr>
<tr>
<td>RC2</td>
<td>$K_{s_d}$</td>
<td>Monod half-saturation constant for electron donor</td>
<td>0.5</td>
<td>[M/L$^3$]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC3</td>
<td>$K_{s_a}$</td>
<td>Monod half-saturation constant for electron acceptor</td>
<td>0.5</td>
<td>[M/L$^3$]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC4</td>
<td>$Y_{x/d}$</td>
<td>Biomass produced per unit of electron donor utilized</td>
<td>0</td>
<td>[-]</td>
<td>Const</td>
<td>n/a Stoichiometry</td>
</tr>
<tr>
<td>RC5</td>
<td>$Y_{a/d}$</td>
<td>Electron acceptor used per unit of electron donor utilized</td>
<td>0</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
<tr>
<td>RC6</td>
<td>$K_{\text{decay}}$</td>
<td>First-order bacterial death or decay rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-5(Rates)</td>
</tr>
<tr>
<td>RC7</td>
<td>$K_{\text{attach}}$</td>
<td>First-order bacterial attachment rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC8</td>
<td>$K_{\text{detach}}$</td>
<td>First-order bacterial detachment rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
</tbody>
</table>

*Note: This reaction module describes a general double Monod model. By setting appropriate yield and kinetic constants, users can model any type of biological systems. Kinetic constants for an aerobic system are given in Clement et al. (1998), and for an anaerobic denitrifying system are given in Clement et al. (1997). Also see Taylor and Jaffe (1990); Hornberger et al. (1992); Zysset et al. (1994); and Reddy and Ford (1996).*

### Sequential First-Order Decay

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>$K_{A}$</td>
<td>PCE first-order degradation rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R3-6(Rates)</td>
</tr>
<tr>
<td>RC2</td>
<td>$K_{B}$</td>
<td>TCE first-order degradation rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC3</td>
<td>$K_{C}$</td>
<td>DCE first-order degradation rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC4</td>
<td>$K_{D}$</td>
<td>VC first-order degradation rate</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC5</td>
<td>$Y_{tce/pc}$</td>
<td>Yield coefficient, TCE/PCE</td>
<td>0.792</td>
<td>[-]</td>
<td>Const</td>
<td>n/a Stoichiometry</td>
</tr>
<tr>
<td>RC6</td>
<td>$Y_{dce/tce}$</td>
<td>Yield coefficient, DCE/TCE</td>
<td>0.738</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
<tr>
<td>RC7</td>
<td>$Y_{vc/dce}$</td>
<td>Yield coefficient, VC/DCE</td>
<td>0.644</td>
<td>[-]</td>
<td>Const</td>
<td></td>
</tr>
</tbody>
</table>

*Note: Yield values are on a mg/L basis; to be consistent, the user must use mg/L units for all concentrations when using this reaction module.
Aerobic/Aerobic PCE/TCE Dechlorination

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Caption</th>
<th>Description</th>
<th>Default</th>
<th>Units</th>
<th>Constant / Spatially Variable</th>
<th>Node in Model Explorer</th>
</tr>
</thead>
<tbody>
<tr>
<td>RC1</td>
<td>Kp0</td>
<td>Sequential (anaerobic) reaction rate for PCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td>R-3(Rates)</td>
</tr>
<tr>
<td>RC2</td>
<td>Kt1</td>
<td>Sequential (anaerobic) reaction rate for TCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC3</td>
<td>Kt2</td>
<td>Aerobic decay rate for TCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC4</td>
<td>Kd1</td>
<td>Sequential (anaerobic) reaction rate for DCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC5</td>
<td>Kd2</td>
<td>Aerobic decay rate for DCE</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC6</td>
<td>Kv1</td>
<td>Sequential (anaerobic) reaction rate for VC</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC7</td>
<td>Kv2</td>
<td>Aerobic decay rate for VC</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC8</td>
<td>Ke1</td>
<td>Sequential (anaerobic) reaction rate for ETH</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
<tr>
<td>RC9</td>
<td>Ke2</td>
<td>Aerobic decay rate for ETH</td>
<td>0</td>
<td>[1/T]</td>
<td>Const/Var</td>
<td></td>
</tr>
</tbody>
</table>

*Note*: All the yield values are fixed internally; to be consistent, use mg/L units for all species concentrations.

**Species Parameters**

The Species Parameters include the Sorption and Reaction parameters used by the selected transport settings. The available parameters will depend on what sorption and reaction settings you selected in the Modeling Objectives workflow step. The parameters presented in the Species Parameters Database window are from the parameters listed in the Species Parameters Tab in the Modeling Objectives workflow step.

If no sorption or reactions are selected in the current transport variant, then no sorption or reaction parameters are required for the simulation, and there will not be an option for “Species Parameters” at the Define Properties workflow step.

The parameters are described by [retardation model](#) and [reaction model](#) below.

**Retardation Model - Species Parameters**

The list of parameters required for each species based on each retardation model is provided below, associated base units are provided in square brackets [ ], where L signifies length (e.g. meters, feet), M signifies mass (e.g. kilograms, pounds), and T signifies time (e.g. seconds, days, years).

A tabulation of which retardation model is supported by each of the transport engines is provided in the Define Modeling Objectives workflow step. For more detailed documentation
on each of the retardation model parameters, please refer to the documentation of the relevant transport engine (i.e. MT3DMS, RT3D, SEAWAT, or MODFLOW-SURFACT).

**Linear Isotherm (equilibrium controlled)**
- $K_d$ is the distribution coefficient $[1/(M/L^3)]$

**Freundlich (equilibrium-controlled)**
- $K_f$ is the Freundlich constant $[1/(M/L^3)^a]$  
- $a$ is the Freundlich exponent [-]

**Langmuir (equilibrium-controlled)**
- $K_L$ is the Langmuir is constant $[1/(M/L^3)]$
- $S$ is the total concentration of sorption sites available [-]

**First order kinetic sorption (non-equilibrium)**
- $K_d$ is the distribution coefficient $[1/(M/L^3)]$
- $K_{mass}$ is the first-order kinetic mass transfer coefficient $[1/T]$

**Dual-domain mass-transfer without sorption**
- SCONCIM is the initial concentration in the distribution coefficient $[1/(M/L^3)]$
- $K_{mass}$ is the first-order kinetic mass transfer coefficient $[1/T]$

**Dual-domain mass-transfer with linear sorption in mobile domain**
- SCONCIM is the initial concentration in the immobile domain $[M/L^3]$
- $K_d$ is the distribution coefficient for the mobile domain $[1/(M/L^3)]$
- $K_{mass}$ is the first-order kinetic mass transfer coefficient $[1/T]$

**Dual-domain mass-transfer (with the same linear sorption in mobile and immobile domains)**
- SCONCIM is the initial concentration in the immobile domain $[M/L^3]$
- $K_d$ is the distribution coefficient for both the mobile and immobile domain $[1/(M/L^3)]$
- $K_{mass}$ is the first-order kinetic mass transfer coefficient $[1/T]$
Dual-domain mass-transfer (with different linear sorption in mobile and immobile domains)

- **SCONCIM** is the initial concentration in the immobile domain \([\text{M/L}^3]\)
- **Kd** is the distribution coefficient for the mobile domain \([1/(\text{M/L}^3)]\)
- **K_mass** is the first-order kinetic mass transfer coefficient \([1/T]\)
- **KdIm** is the distribution coefficient for the immobile domain \([1/(\text{M/L}^3)]\)

Dual-domain mass-transfer with nonlinear sorption in mobile domain

- **SCONCIM** is the initial concentration in the immobile domain \([\text{M/L}^3]\)
- **Kf** is the Freundlich constant for the mobile domain \([1/(\text{M/L}^3)^a]\)
- **a** is the Freundlich exponent for the mobile domain [-]
- **K_mass** is the first-order kinetic mass transfer coefficient \([1/T]\)

Dual-domain mass-transfer (with the same nonlinear sorption in mobile and immobile domains)

- **SCONCIM** is the initial concentration in the immobile domain \([\text{M/L}^3]\)
- **Kf** is the Freundlich constant for both the mobile and immobile domain \([1/(\text{M/L}^3)^a]\)
- **a** is the Freundlich exponent for both the mobile and immobile domain [-]
- **K_mass** is the first-order kinetic mass transfer coefficient \([1/T]\)

Dual-domain mass-transfer (with different linear sorption in mobile and immobile domains)

- **SCONCIM** is the initial concentration in the immobile domain \([\text{M/L}^3]\)
- **Kf** is the Freundlich constant for the mobile domain \([1/(\text{M/L}^3)^a]\)
- **a** is the Freundlich exponent for the mobile domain [-]
- **K_mass** is the first-order kinetic mass transfer coefficient \([1/T]\)
- **KfIm** is the Freundlich constant for the immobile domain \([1/(\text{M/L}^3)^{aim}]\)
- **aim** is the Freundlich exponent for the immobile domain [-]

Reaction Model - Species Parameters
The list of parameters required for each species based on each reaction model is provided below, associated base units are provided in square brackets [ ], where L signifies length (e.g. meters, feet), M signifies mass (e.g. kilograms, pounds), and T signifies time (e.g. seconds, days, years).

A tabulation of which reaction model is supported by each of the transport engines is provided in the Define Modeling Objectives workflow step. For more detailed documentation on each of the retardation model parameters, please refer to the documentation of the relevant transport engine (i.e. MT3DMS, RT3D, SEAWAT, or MODFLOW-SURFACT).

Please Note: The RT3D reactions have specific default species for each of its associated reaction modules. Selecting a reaction module specific to RT3D will reset the species list and all associated species and model parameters.

**First-Order Irreversible Decay**

- **K_mobile** is the first-order decay rate coefficient for dissolved constituents in both the mobile and immobile domain [1/T]

- **K_sorbed** is the first-order decay rate coefficient for sorbed constituents in both the mobile and immobile domain [1/T]

Please Note: First-order decay rate coefficients can be derived from half-life values, which are more commonly available. Concentrations that follow first-order decay can be shown to change over time by the equation:

\[ C_t = C_0 e^{-kt} \]

where:

- \( C_t \) is the concentration of the constituent at time \( t \) [M/L^3]
- \( C_0 \) is the initial concentration of the constituent [M/L^3]
- \( k \) is the first order decay rate [1/T]
- \( t \) is elapsed time [T]

When the constituent concentration reaches half of its initial concentration (i.e. \( C_t = 0.5 \times C_0 \)), the equation above can be rewritten as:

\[ k = \ln 2 / t_{1/2} \]

where:

- \( k \) is the first order decay rate [1/T]
\( t_{1/2} \) is the half-life of the constituent C [T]

**Zeroth-Order Irreversible Decay**

- **K\textsubscript{mobile}** is the zeroth-order decay rate coefficient for dissolved constituents in both the mobile and immobile domain [1/T]
  
- **K\textsubscript{sorbed}** is the zeroth-order decay rate coefficient for sorbed constituents in both the mobile and immobile domain [1/T]

**RT3D Reaction Modules**

When selecting one of the reaction modules associated with RT3D:

- Instantaneous aerobic degradation of BTEX;
- Six-Species, First-Order, Rate-Limited, BTEX Degradation using Sequential Electron Acceptors;
- Rate-Limited Sorption;
- Double-Monod Model;
- Sequential First-Order Decay; or
- Aerobic/Anaerobic PCE/TCE Dechlorination

The **Absolute Tolerance (ATOL)** and **Relative Tolerance (RTOL)** species parameters will be added for each species. These tolerance parameters are used by the differential equations solver in RT3D to control convergence errors while solving the applicable reaction model. Setting ATOL(i) = 0.0 results in a pure relative error test on Species i, while setting RTOL(i) = 0.0 results in a pure absolute error test on Species i. For practical problems, the following rules of thumb may be used to set ATOL and RTOL values:

- set RTOL(i) = 10\(^{-m+1}\), where \( m \) is the desired number of significant digits for concentration output for species i (\( C_i \))
- set ATOL(i) to a small value at which the absolute value of \( C_i \) is essentially insignificant (typically between \( 10^{-6} \) to \( 10^{-9} \))

⚠️ **Please Note**: that RTOL + ATOL > 0

---

**Warning!** Actual (global) errors may exceed the local tolerances, so choose ATOL(i) and RTOL(i) conservatively.
**Longitudinal Dispersion**

Dispersion is a physical process that tends to ‘disperse’, or spread, the contaminant mass in the X, Y and Z directions along the advective path of the plume, and acts to reduce the solute concentration. Dispersion is caused by the tortuosity of the flowpaths of the groundwater as it travels through the interconnected pores of the soil.

Dispersion is calculated using the equation:

\[
D = \alpha_L \times \frac{V_L^2}{|v|} + \alpha_H \times \frac{V_H^2}{|v|} + \alpha_V \times \frac{V_V^2}{|v|} + D^* 
\]

where:

- \(D\) is the Dispersion Coefficient [L^2/T]
- \(\alpha_L\) is the longitudinal dispersivity [L]
- \(V_L\) is the longitudinal velocity of flow along the plume migration pathway [L/T]
- \(\alpha_H\) is the horizontal dispersivity [L]
- \(V_H\) is the horizontal velocity of flow along the plume migration pathway [L/T]
- \(\alpha_V\) is the vertical dispersivity [L]
- \(V_V\) is the vertical velocity of flow along the plume migration pathway [L/T]
- \(D^*\) is the diffusion coefficient [L^2/T]
- \(|v|\) is the magnitude of seepage velocity [L/T]

MT3DMS, RT3D, SEAWAT, and SURFACT calculate the Dispersion Coefficient for the mass transport model using the following parameters:

- Longitudinal Dispersivity for each transport grid cell
- Ratio of Horizontal to Longitudinal Dispersivity for each layer
- Ratio of Vertical to Longitudinal Dispersivity for each layer
- Molecular Diffusion Coefficient for each layer

Longitudinal Dispersion can be defined using the regular set of tools provided for most parameters (i.e. cell-by-cell, by layer/row/column, or using polylines/polygons/data objects). The Longitudinal Dispersion can also be defined on a layer-by-layer basis by right-clicking the Longitudinal Dispersion object in the model tree and selecting the Dispersion Parameters option.
Longitudinal Dispersion can also be defined based on property zones by right-clicking the Longitudinal Dispersion object in the model tree and selecting the Edit Property Zones... option. The absolute values of the horizontal and vertical dispersion parameters will be adjusted according to the property zone values.
For SEAWAT models, you can specify a diffusion coefficient for each species using the MDCOEFF variable at the define modeling objectives step. The values will be used if the Multi-diffusion option is set to true in the Advanced Settings for the DSP package in the Translation step.

10.5 Define Boundary Conditions

At the Define Boundary Conditions step, you can define and edit the various boundary conditions for your model. This can be done in Layer view as well as in the Row or Column views.

 располагается внизу страницы.

Please Note: Defining and Editing is not supported in the 3D view of finite difference models.

- Under the Toolbox, use the combo box to select from the various Boundary condition types.
- Use the same tools as described in the previous step to manipulate the views.
Each boundary condition group of cells will appear as its own node in the model tree. You can right click on this to adjust style settings. In addition you can load any group of boundary condition cells into a stand-alone 3D Viewer.
To delete a specific boundary condition cell group, right-click on this node in the tree, and select Delete.

To validate there are no warning or errors associated with a boundary condition, right-click a specific boundary condition and select Validate... and if there are any errors or warning you will be provided an option to resolve the issue as shown in the dialog below. This is available for Constant Head, River, General Head, and Drain. The validation process will check to see if the attributes of the boundary condition will place it below the bottom of the cell you assigned it to. Then it will give you options to correct this. If you choose to Ignore the errors you may encounter errors when trying to translate and run your model.
In general, each of the boundary condition types has the same set of graphical tools on the left-hand toolbar for editing the model boundary conditions. The following is a description of how to use these options. For specific descriptions of the data requirements and data entry options, refer to the sections describing each boundary condition type.

**Import**

The import button allows you to import boundary condition text files in IJK format. Simply browse to the file and select Open to import the boundary condition. This option is useful when making edits to the boundary condition as described in Edit Attributes in TXT Format.

**Assign**

The Assign button allows you to assign new boundary conditions. You have options to assign by Polyline or Polygon, by Using Data Object or by Cells. These options are described further below.

- **Polygon** or **Polyline**: Use this option to digitize one or more shapes and assign Boundary Condition attributes to the respective shapes. Use the left mouse button to start drawing and add attributes at the desired locations. Once you have finished digitizing, right click with the mouse to close the shape and select "Define Attributes" as shown below.
• **Using Data Object**: Use this option when you have a polyline or polygon data object already imported into the projects, and you would like to use this shape to define the geometry.

• **Cells**: Use this option when you want to select individual cells to assign a boundary condition. For Recharge and Evaporation this can be useful to change only a few cells from one zone to another - as the Define Boundary Condition dialog allows you to enter zone numbers.

The Define Boundary Condition attributes window will appear as shown below.
Provide the desired Name and Description.

Select Next to proceed.

Define the attributes using the various methods; click Finish when you are done.

This dialog allows you to copy and paste from Excel to make defining (or editing) the attributes easy.
The following buttons are available:

<table>
<thead>
<tr>
<th>Button</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Assign to current cell (F1)" /></td>
<td>Assign to current cell (F1)</td>
</tr>
<tr>
<td><img src="image" alt="Assign to column (F2)" /></td>
<td>Assign to column (F2)</td>
</tr>
<tr>
<td><img src="image" alt="Apply to selected cells (F3)" /></td>
<td>Apply to selected cells (F3)</td>
</tr>
<tr>
<td><img src="image" alt="Interpolate values between selected cells" /></td>
<td>Interpolate values between selected cells</td>
</tr>
<tr>
<td><img src="image" alt="Toggle automatic calculation of calculated fields" /></td>
<td>Toggle automatic calculation of calculated fields (e.g. river length x river width x river conductivity vs. explicit conductance)</td>
</tr>
</tbody>
</table>

The interpolate option allows you to select 2 cells (rows) and then when you select Interpolate - it will calculate the values for the cells (rows) in between using linear interpolation.

For more details on the options for defining attributes, see Define Boundary Conditions (Attributes).

If your boundary condition is transient switch the schedule drop down to transient to enter your schedule.

You can copy and paste your time schedule from Excel into this dialog to make defining your boundary condition quicker and easier.

By default the Assign to appropriate layer is selected. This means if you enter values for the attributes of a boundary condition (for example the Starting Head) which would place it into another layer - this will occur automatically once you save your boundary condition.

**Please Note**: When digitizing boundary conditions, Visual MODFLOW Flex will automatically assign these to the appropriate layer, based on the attributes you define.
For more details on creating a Wall (HFB) Boundary Condition, please see Define Wall (HFB).
For more details on creating Wells Boundary Condition object, please see Define Pumping Wells.

**Edit**

When you select the Edit... button you must select a cell containing the boundary condition you wish to edit. The following dialog will appear (similar to defining the boundary condition).

![Edit boundary condition dialog](image)

The attributes for each cell containing the boundary condition are displayed and can be edited. The cell you selected will be highlighted in the dialog. If you select other rows in the dialog - the corresponding cell will be highlighted in the viewer.

By default the Assign to appropriate layer is selected. This means if you adjust the attributes of a boundary condition (for example the Stage) which would place it into another layer - this will occur automatically once you save your edits.

💡 To view the attributes for Recharge or Evapotranspiration zones, see the Database section below.

**Script**

If preferred, you can make your boundary condition edits by using the Script option. On this tab you will find the script for any edits you make on the Edit cells tab allowing you to learn the script language. You can re-use the script to "replay" your edits to your boundary conditions.
You can also build expressions using the Expression Builder.

Erase

The Erase button allows you erase the boundary condition assigned to either a single cell or a group of cells. This option is available for all boundary conditions (except Recharge and Evapotranspiration) as well as Walls and Pumping Wells.

Copy

The Copy / Layer/Row/Column option is used to copy all of the boundary data from a selected boundary group to one or more selected model layers, rows, or columns (based on the active view). This option is available for: Constant Head, River, General Head, Drain, Specified Flux and Constant Concentration. When this option is launched, you will see only those boundary condition groups that are in the currently selected layer. Please use caution when copying boundary conditions for an LGR model, as you cannot copy boundary conditions from layers in a parent grid to layers in a child grid; you can only copy boundary conditions within the grid in which that it lies (within parent grid layers/rows/columns or within child grid layers/rows/columns).
Database

The Database button becomes active when you select either Recharge or Evapotranspiration for the Boundary Condition type. This will load the zone database window as shown in the following figure.
In this grid, you can modify the values for existing zones.

**Specifying Active/Inactive Time Periods**

For Constant Head, River, Drain, General-Head, and Specified Flux, you can disable the selected Time Periods by removing the check marks from the Active column (see below), which “turns off” the boundary condition for that particular Time Period. The corresponding package will have no attributes defined for those cells covered by that boundary condition, for that specific stress period. For Recharge and Evapotranspiration zones, you can define a zero rate for those cells in those time periods.

**Editing Well Attributes**

To edit wells,

- Select "Well" from the Boundary Condition list.
- Be sure you are in the correct layer where your well is screened.
- Select [Edit ->] Single from the toolbox.
- Select the desired well cell.
- You will see a window similar to the one below:
Please Note: if you make edits to the well pumping rates in this screen, these changes are not propagated back to the raw (original) wells data object in the Data Explorer.

Display
By default, Visual MODFLOW Flex uses the following color scheme for Boundary Condition Cells

<table>
<thead>
<tr>
<th>Boundary Condition Type</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant Head</td>
<td>Red</td>
</tr>
<tr>
<td>River</td>
<td>Blue</td>
</tr>
<tr>
<td>General Head</td>
<td>Green</td>
</tr>
<tr>
<td>Drain</td>
<td>Grey</td>
</tr>
<tr>
<td>Lake</td>
<td>Light blue</td>
</tr>
<tr>
<td>Recharge</td>
<td>Varies (zone based)</td>
</tr>
<tr>
<td>Evapotranspiration</td>
<td>Varies (zone based)</td>
</tr>
</tbody>
</table>
### Unsaturated Zone (UZF)

<table>
<thead>
<tr>
<th>Feature</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stripped</td>
<td>Varies (zone based)</td>
</tr>
<tr>
<td>Seepage Face</td>
<td>Bright Blue</td>
</tr>
<tr>
<td>Specified Flux</td>
<td>Beige</td>
</tr>
<tr>
<td>Time Varying Material</td>
<td>Orange</td>
</tr>
<tr>
<td>Wall (Horizontal Flow Barrier)</td>
<td>Olive</td>
</tr>
<tr>
<td>Wells</td>
<td>Light red</td>
</tr>
</tbody>
</table>

### Recharge and Evapotranspiration

Recharge and Evapotranspiration have additional settings for rendering the cells by either ZoneID or Rate (either Recharge or Evapotranspiration). To access these settings, right-click on the Recharge (or Evapotranspiration) node in the model tree and select Settings. The following dialog will appear:
From the tree on the left side of this window, select Colors under Style/Recharge. In the main window, under "**Attribute**" you can then change this between Zone or Rate; as you change this, the color range in the grid will refresh. Upon clicking Apply or OK, the 2D/3D views will be refreshed.

**Use Default Conductance**

For Drain, River, General Head, and Lake you will find an options to Use Default Conductance within the Define Boundary Condition dialog.

This allows you to enter attributes more easily measured and have the conductance calculated for you. For example, when assigning a River you need to provide the River width, Riverbed thickness and Riverbed Conductivity - and the Conductance will be calculated for you.
If you unselect the Use default conductance button you will then need to provide the Conductance values yourself.

**Exporting Boundary Conditions**
The boundary condition cell locations (and attributes) be exported to shapefile; see Export for more details.

**Adjusting Style Settings**
The style settings of some of the Boundary condition cells

Right click on the desired Boundary condition cell group from the model tree and select `Settings`.

The Settings dialog will appear; the settings are similar to described in Points.
10.5.1 Edit Attributes in TXT Format

In certain cases, it may be helpful to export your boundary condition to a text file and make edits to the exported file and then import the boundary condition back into Visual MODFLOW Flex.

You can find this option by right clicking on your individual boundary condition in the Model Explorer:

![Model Explorer screenshot](image)

You will be provided with a dialog to save the boundary condition as a .txt file. When working with regular (finite difference) grids, the format will be I,J,K while working with USG grids it will be ordered by ID.

You can then make your edits within the .txt file and when you are done import the boundary condition back into Visual MODFLOW Flex.

You will find the import option within the Numerical Workflow at the Define Boundary Condition step.
Be sure to select the correct boundary condition type from the drop down before attempting to import the boundary condition file.

10.5.2 Transport
If you are used to working with Visual MODFLOW Classic, you will notice a difference in how transport boundary conditions are handled in Visual MODFLOW Flex. In Visual MODFLOW Classic, transport boundary conditions were defined separately from the flow boundary conditions using the types Constant Concentration, Recharge Concentration, Evapotranspiration Concentration, and Point Source. In Visual MODFLOW Flex, the sink/source parameters for transport models (which are time and species concentrations) are defined as part of flow boundary conditions, which is a more natural representation. This means you do not define separate cell geometries for transport boundaries, you simply define species concentrations while defining the flow boundary conditions, where required. Constant Concentration is an exception to this rule, since it does not need to coincide with a prescribed flux, you will still see a "Constant Concentration" boundary condition type, allowing you to define the geometry (cells) and parameters (time and species concentrations) for this Boundary Condition type.

When Transport is active in your model run, and you define a new boundary condition, you will see parameters for Species Concentration as part of the Boundary Condition attributes (eg. Conc001, Conc002, etc.). These will have a default value of -1, indicating that no mass sink/source is defined for this group of boundary condition cells. As soon as you change this value to 0 or greater, then these cells will be treated as sinks/sources.

**Time Schedule:**
Currently in Visual MODFLOW Flex when you import a Visual MODFLOW Classic model with point source concentrations, the program assumes that the time schedule for the point sources are the same as that of the flow boundary condition to which it is assigned; if it is necessary to have a different time schedule, then please re-define the flow boundary condition with the necessary time adjustments, then re-assign the point source concentrations.

**Point Source assigned to Cells instead of Well Screens:**
In Visual MODFLOW Classic point source concentrations were assigned to cells, and not to the screened interval. In this fashion, it is possible to have different point source concentrations over the screened interval (over different model layers). This configuration is not supported in Visual MODFLOW Flex. When you import a Visual MODFLOW Classic model and this scenario is encountered, you will receive a warning message of the offending wells. Please fix/verify these wells before proceeding.

**Multiple Wells in the Same Cell:**
In Visual MODFLOW Classic, multiple wells could belong to a cell, however point source concentrations were assigned to a cell. In Visual MODFLOW Flex, point source concentrations are assigned directly to the well screen, which is a more natural approach. If you import a model from Visual MODFLOW Classic, and it detects that there are multiple wells in a cell that also contains a point source concentration, you will receive a warning during import with a list of the offending wells. Please fix/verify these wells before proceeding.
10.5.3 Wells

Pumping Wells

Visual MODFLOW Flex provides an improved approach and interface for working with pumping wells in the conceptual and numerical model. Visual MODFLOW Flex follows a logical approach of defining the well rate over the entire well, and independent of the numerical grid. With this "conceptual" approach, a single well can contain multiple screens, fully penetrating screens, or partially penetrating screen intervals over the depth of the wellbore. Pumping wells are defined at X,Y co-ordinates, and not assigned to specific grid cells. This provides flexibility in changing the numerical grid and the ability to readily use well locations (points) as grid node generators for MODFLOW-USG.

Pumping well data are shared between the conceptual and numerical models so that modifications to the conceptual model will translate into the numerical model. Pumping wells are presented in an easy-to-use view showing the well location, screen geometry, and pumping rates. Since the rate is assigned to the entire well, you don't need to worry about figuring out how much discharge is proportioned to each model layer; Visual MODFLOW Flex does this calculation for you automatically. For details on how well rates are calculated for the .WEL package, see the section Calculation of Well Rates.

In addition, if you are running a transport simulation, point source concentrations for an injection well are defined/edited on a per well level, an approach that more closely represents actual field conditions.

Further improvements to wells that will enhance the “ease-of-use" include:

- Import pumping schedule data in relative times (starting at 0 days), in addition to absolute dates (eg. 04/22/2015)
- Setting individual wells as active/inactive in the database view, and having this setting translated to the .WEL package
- Adding one or more new wells with the click of a mouse
- Enhanced visualization of the well screen in cross-section view (by row/column), and in 3D view
- Flexible labeling and rendering options in 2D and 3D.
- Display the well name in the .WEL package, to correlate the conceptual well to a specific MODFLOW grid cell.

Working With Wells in Projects Prior to v.2015.1

Visual MODFLOW Flex is backwards compatible with projects made in v.2014.2 and older. When you open such a project that contains pumping wells in the numerical model, and try to edit a well cell, you will see the following window:
You may proceed with editing the well rates/schedule for wells in the numerical model. However, you are strongly encouraged to convert wells in this old (cell-based) format into the new wells (conceptual object) format. The benefits of working with the new (conceptual) wells are the following:

- more logical editing of the well rates/schedule defined over the screen interval (as opposed to defining rates for individual cells)
- improved display of the wells (in the conceptual model and numerical model, in 3D, layer and cross-section view)
- robust translation: accommodate multiple wells in the same cell, with individual entries in the .WEL package
- option to set individual wells as active or inactive for the simulation.
- simple option to point-and-click to create a well in the numerical model

The process of conversion is described below.

**Converting From A Cell-Based (Old) Wells Object To Conceptual (New) Wells Object**
Before You Start!

The conversion process will involve deleting the wells in the numerical model and conceptual model and creating a new wells object utilizing the "raw wells" object (from the Data tree). If you have done cell-based editing of well rates in the numerical model, these edits will be lost. In addition, if you have assigned point source concentrations to wells in the numerical model (for a transport scenario), these changes will also be lost (you will need to re-assign these parameters after creating the new wells object).

If you have made significant edits to well rates/schedules in the numerical model, then you may wish to avoid this conversion.

Before proceeding:

- Make a backup copy of your project
- Make a list of the changes you made to the numerical wells object; you may wish to export out the well attributes to a CSV file (right-click on Wells in the numerical model and select Edit Attributes; save this file as a separate XLS or .CSV file), and use this as a means of checking your new wells later on.
- Once you have finished, cross-check the wells in the conceptual and numerical model to verify your data are correct, and if necessary, adjust the rates/schedules.
- If your previous wells object contained point source concentrations, then re-assign these values to the new wells object.

Deleting Old Wells Objects
- In a Numerical Model
  - Open the project and expand the Model Explorer
  - Locate the Wells object, under Run/Inputs/Boundary Conditions/Wells
  - Right click on this object and select "Delete"
  - Click Yes to proceed when the warning message appears.
  - Proceed to the next section to also delete the Conceptual Wells from the same project.

- In a Conceptual Model
  - Open the project and expand the Model Explorer
  - Locate the Conceptual Pumping Wells object, under Simulation Domain / Model Domain / Boundary Conditions / Wells
  - Right click on this object and select "Delete"
  - Click Yes to proceed when the warning message appears.

**Please Note:** deleting the conceptual pumping wells object will also remove the corresponding numerical wells from any numerical models that were derived from that conceptual model.

Creating a New Wells Object

In a Numerical Model
  - Select the Numerical Model workflow tab (from the list of active viewers/workflows across the top of the main window)
  - Go to the Define Boundary Conditions step
  - Under Toolbox, choose "Wells" from the list of available boundary condition types
  - Select Assign / Using Data Object; as shown below:

![Wells Toolbox](image)

- The following window will appear:
- Enter a name for the wells
- On the Data tree, locate the "raw" pumping wells object that you imported into the project
- Click once on the caption for this well to select it.
- Drag the wells into the empty text box under "Select Raw Wells Object"
- Release the mouse button
- The list of wells will appear in the window; an example is below
• Review the well data, then click OK to complete the process.
• A new item will appear on the Model Explorer under Run/Inputs/Boundary Conditions/Wells, and the new wells should appear in the Flex viewer (note if you are viewing by layer, you may need to scroll to the appropriate layers to see the new wells in those screened intervals)

Please Note: Creating a new wells object will also create a corresponding Conceptual Model Pumping Wells object; the data for these two wells are the same.

• Once the new wells object is created, you can edit the attributes by selecting Edit/Well from the toolbox, and select any well in the numerical model.
• Verify that the new wells object contains the correct values, and if necessary make the appropriate corrections. If your previous wells object contained species concentrations, then re-assign these now.

In a Conceptual Model

Please Note: This step is only necessary if you have just a conceptual model with no numerical models.
Select the Conceptual Workflow tab (from the list of active viewers/workflows across the top of the main window)

Go to the Define Boundary Conditions step (as shown below)

Select Define Pumping Wells. A Create Well Boundary Condition window will appear; the layout is identical to what is shown in the section above for creating wells in the Numerical Model.

Enter a name for the wells

On the Data tree, locate the "raw" pumping wells object that you imported into the project

Click once on the caption for this wells object to select it.

Drag the wells into the empty text box under "Select Raw Wells Object".

Release the mouse button

The list of wells will appear in the window

Verify that the new wells object contains the correct values, and if necessary make the appropriate corrections. If your previous wells object contained species concentrations for a transport run, then re-assign these now.

Click OK to complete the process.

A new item will appear on the Model Explorer under Simulation Domain / Model Domain / Boundary Conditions / Wells
Once the new wells object is created, you can edit the attributes by right-clicking on the Wells object from the Model explorer under Simulation Domain / Model Domain / Boundary Conditions and selecting Edit Boundary Condition from the menu as shown below.

Options for Wells in the Numerical Model

When you are working in the numerical model, there are two options for adding wells to your model

Assign / Wells

Allows you to point and click at locations in the grid (in the current layer) to define one or more well locations. This option is useful if you want to add just a few wells with a simple schedule, and do not have this data already in an Excel file.

- Select the desired model layer where you want to add a well
- Choose Assign/Wells
- Using the left mouse button, point and click at the desired location.
- When you are finished, click on the "Finish" button on the Toolbox (this can be found under the "Assign" option), or right-click anywhere in the layer view of the grid and select "Define Attributes".
- In the case where a wells object already exists, the following window will appear: (if there are no wells in the project, then the “Edit Wells” window will appear as explained below.
The newly created wells can be added to an existing wells object (new conceptual wells only!) or you can add these wells to a new wells group. This scenario is useful if you want to categorize and distinguish wells in your model (for example separate injection or pumping wells, private vs. public supply wells, etc.)

- Click OK
- The "Edit Wells" window will appear; in this window you can define the screen geometry and pumping schedule. When you add a well with the click of a mouse, the default screen that is created will be fully over the layer thickness; the well Z max will be top of layer 1 and the well Z min will be the bottom of the bottommost layer.
- Click OK to finish creating the wells.

**Assign / Using Data Object**

Allows you to create a pumping wells object by using an imported wells object as the data source.

- Select the Numerical Model workflow tab (from the list of active viewers/workflows across the top of the main window)
- Go to the Define Boundary Conditions step
- Under Toolbox, choose "Wells" from the list of available boundary condition types
- Select Assign / Using Data Object; as shown below.

- The following window will appear:
- Enter a name for the wells
- On the Data tree, locate the “raw” pumping wells object that you imported into the project
- Click once on the caption for this well to select it.
- Drag the wells into the empty text box under "Select Raw Wells Object"
- Release the mouse button
- The list of wells will appear in the window; an example is below
• Review the well data, then click OK to complete the process.

**Accommodating Wells with Different Start Times**

If you choose a wells data object that contains wells that start at different times, then you will receive a warning when attempting to use this object to create pumping wells; an example is shown below:
The following validations are done:

1. The pumping well start date must occur on or after the model start date; any wells with a pumping schedule before the start date will be truncated (cut-off).
2. All wells must start at the same start date, and this must be the model start date; in practice, this may not always occur, so VMOD Flex will automatically insert pumping rate of 0 for all wells that do not start at the model start date.

**Edit Wells**
- Browse to a layer that contains a well; it will appear as a point (colored pink)
- Select Edit/Single
- Click on a Well in the layer view
- The Edit Wells window will appear as explained above.
- Make the desired edits then click OK to apply the changes.

Wells can be set as active/inactive for the simulation; use the "Active" column in the Wells grid to set the appropriate option; wells that are set as Inactive will not be translated to the .WEL package.

**Transport Parameters (Species Concentrations)**
When you have Transport set as active in the Modeling Objectives, you will have the option to define species concentrations to injection wells; this can be done in the Pumping Schedule, under the "Conc001" column. The injected concentration is assigned to the entire well interval. The default value of -1 is a flag indicating that the injected water does not contain the associated species. One column will be added to the pumping schedule for each species defined in the Modeling Objectives step - the column header will correspond to the species name.
Export
The pumping wells can be exported to a comma separated value (CSV) file; right click on the Model Explorer and select "Export"; a CSV file will be generated which can be loaded into a text editor or Excel.

10.5.4 Theory

A discussion on how boundary conditions are conceptualized and parameterized in Visual MODFLOW Flex can be found here.

Steady-State vs. Transient Flow Boundary Conditions

For transient simulations, MODFLOW requires the time element of the boundary conditions to be defined using Stress Period “counters” as opposed to using “real” times. As a result, each time interval for a transient model must be determined in terms of Stress Periods before any boundary condition data is defined. Unfortunately, accommodating this format is quite tedious because the data collected for parameters such as rainfall and groundwater recharge don’t always follow the same time schedules as data collected for other boundary conditions, such as well pumping rates and surface water levels. This approach also makes it difficult to utilize raw field data collected and recorded in terms of real times.

In Visual MODFLOW Flex, a Time Period is similar to a Stress Period, but with two important exceptions:

1. A Time Period is defined using real times and real time units, and
2. Each boundary condition grid cell may contain different Time Periods

The advantage of this approach is the ability to clearly see the magnitude of time for each Time Period (as opposed to interpreting data such as “from Stress Period 1 to Stress Period 2”), and it facilitates more convenient methods for importing raw data from different boundary condition types.

Each group of boundary condition grid cells requires a minimum of one Time Period of data containing a Start Time, a Stop Time, and a complete set of data for the selected boundary condition type (the required data for each boundary condition type are described later in this section). For steady-state simulations, Visual MODFLOW Flex requires data for only a single Time Period, while for transient simulations, Visual MODFLOW Flex can accommodate an unlimited number of Time Periods.

For steady-state simulations, the Stop Time value is irrelevant because the term “steady-state” indicates that the model results are not changing with time. Therefore, a Stop Time value of 1 time unit is commonly used. However, if the model is going to be used to evaluate a transient simulation in the future, it is probably a better idea to give it a more realistic value corresponding to the potential time frame of interest.

If a steady-state simulation is run using a model containing transient boundary condition data, only the data from the first Time Period of each grid cell will be used for the steady-state conditions. Furthermore, if the boundary condition consists of starting and ending values (e.g. starting head and ending head for constant head boundaries), the starting value will be used.
Incomplete Boundary Condition Schedules

If the defined period for a boundary condition in a transient model is of a shorter duration that the model itself, Visual MODFLOW Flex will assume that the boundary condition does not exist for the remaining duration. For example, a transient model is run for 10 years, and the recharge boundary condition is defined only for a period up to 7 years, Visual MODFLOW Flex will assume this boundary condition does not exist (i.e. recharge = 0) for the remaining 3 years of the simulation.

The exceptions to this rule are the Constant Head and Constant Concentration Boundaries, which must be defined for the entire simulation.

Please Note: For a steady-state simulation, a minimum of one active grid cell in the model MUST contain a head-dependent boundary condition type. Otherwise, the model is indeterminate and the solution will not converge. This head-dependent boundary condition acts as a reference head for all calculations. The head-dependent boundary condition type can be one or more of the following:

- Constant Head (CHD)
- River (RIV)
- General-Head (GHB)
- Lake (LAK)

For a transient simulation, the specified initial heads are sufficient for a determinant solution.

10.6 Define Observations

Observation wells can be added to the numerical model as described below.

First, import the observation well data as described in the Import Wells section. The raw observation well data can be viewed/edited in the Data Table section.
Once you have the head observation data object imported, navigate to the Define Observations step in the numerical model workflow.

Below the toolbox, you will see the insert blue arrow; select your Observation Wells data object from the data explorer and click on the button.

The observation wells should then appear in the 2D view and will also appear as a new node on the numerical model tree on the Model Explorer.

⚠️ Please Note: If you add Observations that are defined in Absolute times, and those times are before the model start date defined in the modeling objectives, then you will receive a warning message during creation. These head/concentration observations cannot be included in the numerical model and will be filtered out.

**Observation Groups**

If you assign subsequent Observation Wells data objects from the data explorer will be prompted with a choice to append the observations to an existing well group or create a new well group.
The head and concentration observation data can be viewed/modified in a CSV format (once it has been created)

- Right-click on the Head (or Concentration) Observations node from the Model explorer
- Select “Edit Attributes...” as shown below.
The data contained inside the Observations data object will be displayed in a .CSV file; this should appear in either Notepad or Excel depending on your system configuration; an example is below:

```
<table>
<thead>
<tr>
<th>Well Name</th>
<th>X(f)</th>
<th>Y(f)</th>
<th>Obs. ID</th>
<th>Obs. Elev</th>
<th>Layer</th>
<th>Obs. Time Head(ft)</th>
</tr>
</thead>
<tbody>
<tr>
<td>OW-1/1</td>
<td>1590.77</td>
<td>2662.82</td>
<td>Observati</td>
<td>268.81</td>
<td>2</td>
<td>228.38</td>
</tr>
<tr>
<td>OW-10A/1</td>
<td>1256.48</td>
<td>2985.59</td>
<td>Observati</td>
<td>215.38</td>
<td>2</td>
<td>230.32</td>
</tr>
<tr>
<td>OW-14/1</td>
<td>1146.97</td>
<td>3884.72</td>
<td>Observati</td>
<td>181.48</td>
<td>4</td>
<td>224.85</td>
</tr>
<tr>
<td>OW-15/1</td>
<td>1060.51</td>
<td>3527.37</td>
<td>Observati</td>
<td>179.14</td>
<td>4</td>
<td>228.05</td>
</tr>
<tr>
<td>OW-15A/1</td>
<td>1037.46</td>
<td>3573.48</td>
<td>Observati</td>
<td>215.16</td>
<td>2</td>
<td>231.85</td>
</tr>
<tr>
<td>OW-16/1</td>
<td>1740.63</td>
<td>1821.32</td>
<td>Observati</td>
<td>215.16</td>
<td>2</td>
<td>220.62</td>
</tr>
<tr>
<td>OW-17/1</td>
<td>1125.68</td>
<td>1694.52</td>
<td>Observati</td>
<td>210</td>
<td>2</td>
<td>221.43</td>
</tr>
<tr>
<td>OW-18/1</td>
<td>1323.91</td>
<td>2351.58</td>
<td>Observati</td>
<td>183.94</td>
<td>4</td>
<td>218.21</td>
</tr>
<tr>
<td>OW-19/1</td>
<td>2359.07</td>
<td>1475.35</td>
<td>Observati</td>
<td>191.79</td>
<td>2</td>
<td>215.92</td>
</tr>
<tr>
<td>OW-1A/1</td>
<td>1844.83</td>
<td>3538.9</td>
<td>Observati</td>
<td>215.16</td>
<td>2</td>
<td>231.13</td>
</tr>
<tr>
<td>OW-2/1</td>
<td>737.75</td>
<td>2432.77</td>
<td>Observati</td>
<td>207.19</td>
<td>2</td>
<td>228.43</td>
</tr>
<tr>
<td>OW-21/1</td>
<td>1440.92</td>
<td>1821.32</td>
<td>Observati</td>
<td>186.21</td>
<td>4</td>
<td>217.67</td>
</tr>
<tr>
<td>OW-24/1</td>
<td>3458.21</td>
<td>680.11</td>
<td>Observati</td>
<td>198.73</td>
<td>2</td>
<td>206.48</td>
</tr>
<tr>
<td>OW-24A/1</td>
<td>3435.15</td>
<td>657.06</td>
<td>Observati</td>
<td>169.85</td>
<td>4</td>
<td>216.21</td>
</tr>
<tr>
<td>OW-26/1</td>
<td>772.33</td>
<td>1821.32</td>
<td>Observati</td>
<td>265.96</td>
<td>2</td>
<td>232.21</td>
</tr>
<tr>
<td>OW-29/1</td>
<td>1763.68</td>
<td>3054.75</td>
<td>Observati</td>
<td>215.16</td>
<td>2</td>
<td>230.18</td>
</tr>
<tr>
<td>OW-31/1</td>
<td>2051.87</td>
<td>2835.73</td>
<td>Observati</td>
<td>267.01</td>
<td>2</td>
<td>227.49</td>
</tr>
<tr>
<td>OW-34/1</td>
<td>1233.42</td>
<td>1072.04</td>
<td>Observati</td>
<td>193.31</td>
<td>2</td>
<td>211.76</td>
</tr>
<tr>
<td>OW-36/1</td>
<td>2236.31</td>
<td>2469.91</td>
<td>Observati</td>
<td>216.98</td>
<td>2</td>
<td>224.87</td>
</tr>
<tr>
<td>OW-3A/1</td>
<td>1406.34</td>
<td>2262.42</td>
<td>Observati</td>
<td>210.98</td>
<td>2</td>
<td>225.93</td>
</tr>
<tr>
<td>OW-4/1</td>
<td>1406.34</td>
<td>2916.42</td>
<td>Observati</td>
<td>220.45</td>
<td>2</td>
<td>229.83</td>
</tr>
<tr>
<td>OW-5/1</td>
<td>1175.79</td>
<td>2745.51</td>
<td>Observati</td>
<td>215.16</td>
<td>2</td>
<td>229.68</td>
</tr>
<tr>
<td>OW-6/1</td>
<td>1337.17</td>
<td>2776.09</td>
<td>Observati</td>
<td>218.43</td>
<td>2</td>
<td>229.73</td>
</tr>
<tr>
<td>OW-7/1</td>
<td>1337.17</td>
<td>2512.96</td>
<td>Observati</td>
<td>212.2</td>
<td>2</td>
<td>228.08</td>
</tr>
<tr>
<td>OW-8/1</td>
<td>1037.46</td>
<td>2347.83</td>
<td>Observati</td>
<td>211.26</td>
<td>2</td>
<td>226.84</td>
</tr>
</tbody>
</table>
```

You can use this interface for numerous types of edits/additions:

- Adding a new observation point at a defined XY location
- Change existing data
- Add a new observation time to an existing observation point.

**Please Note:** the Layer number for the observation point is calculated based on the Obs. Elev; changing the layer number will not change the vertical location of the observation point.

When you are finished with the edits in the .CSV file, you need to save and close this file. Then re-import back the changes into Visual MODFLOW Flex using the steps below:

- Right-click on the Head (or Concentration) Observations node from the Model explorer
• Select “Reload Attributes…”

The modified data will now appear and will be utilized for any charts the next time you complete a translate and run of the model.

10.7 Define Zone Budget Zones

ZoneBudget is a USGS program for computing subregional water budgets for MODFLOW ground-water flow models. Within Visual MODFLOW Flex, you can create new zones for MODFLOW-2000 and -2005 models in the numerical model workflows. If you imported a Visual MODFLOW Classic model, you can also view the zone budget zones that were defined in this numerical model.

At the Define Zone Budget Zones step, you can create new zone budget zones by digitizing new zones using Assign Polyline, Polygon, or Point (Single Cell), or by using 2D/3D point, polyline, or polygon shape objects from the Data tree.

Follow the steps below to create new zone budget zones:

Assign

Polyline

• Use the left mouse button to start digitizing a polyline over the desired cells. Once you have finished digitizing, click on the Finish button from the toolbox, or right click with the mouse and select “Define Attributes”. The following window will appear
Click the New button to assign the selected cells to a New zone; otherwise, you can select to assign cells to an existing zone by selecting the zone from the dropdown list at the top. The selected cells can be assigned to the current layer or multiple layers using the check boxes at the bottom of the window. Once you are finished, click OK.

**Polygon**
- Digitize one or more polygons, and repeat the steps as described above for Polylines.

**Single**
- Left-click on one or more cells and a point will appear. Once finished, right-click and select "Define Attributes", and follow the steps as described above for Polylines.

**Using Data Object....**
- The following window will appear.

- Choose the desired point, polyline, or polygon data object from the tree.
Click on the button (under Select Data Object)
A second window will appear:

Select which zone you want the features to apply to. You have the option to assign the zones to:
- An existing zone
- A new single zone
- Create zones for individual features
- Create zones based on feature attributes

Select the desired model layers where the zone should be applied

A simple example is shown below for a polygon data object (shown in red).
After the zones are generated, they will appear in layer view.

When the selected shape contains multiple polyline or polygon features, then a new zone will be created for each feature.
10.8 Define Particles

Introduction

Particle-tracking is a useful tool for Groundwater Modelers that provides insights and diagnostics in the modeling process. Two particle tracking engines are supported in Visual MODFLOW Flex:

- MODPATH
- MOD-PATH3DU

⚠️ MODFLOW-LGR models are not compatible with the particle tracking in Visual MODFLOW Flex

MODPATH

The computer program MODPATH was developed by the USGS ([Pollock, 1998](#)) to calculate three-dimensional particle tracking pathlines from steady-state and transient flow simulation output obtained using MODFLOW-2000, MODFLOW-2005, MODFLOW-NWT, MODFLOW-SURFACT, and SEAWAT.

MODPATH uses a semi-analytical particle-tracking scheme. The method is based on the assumption that each directional velocity component varies linearly within a grid cell in its own co-ordinate direction. This assumption allows an analytical expression to be obtained describing the flow path within a grid cell. Given the initial position of a particle anywhere in a cell, the co-ordinates of any other point along its path line within the cell, and the time of travel between them, can be computed. Documentation of the theoretical and numerical implementation of MODPATH are given in Pollock (1998), while source code and input documentation are available at the USGS MODPATH webpage. MODPATH in Visual MODFLOW Flex is not currently compatible with MODFLOW-USG or MODFLOW-LGR.

MOD-PATH3DU

The computer program mod-PATH3DU was developed as part of a collaboration between S.S. Papadopoulos & Associates, Inc. (SSPA) and the University of Waterloo (Muffels, et al, 2016) and calculates three-dimensional particle tracking pathlines from steady-state and transient flow simulation output.

Similar to MODPATH, mod-PATH3DU uses a semi-analytical particle-tracking scheme based on the linear interpolation of the velocity field within a given grid cell. The major difference is that mod-PATH3DU supports intra-cell velocity field interpolation using the Waterloo method, which is based on a high-order Runge-Kutta scheme and is valid for any prismatic cell geometry. In addition to the flow engines supported above, mod-PATH3DU also supports MODFLOW-USG.

A more thorough explanation of the method is documented by Rhamadhan (2016).
NOTE: mod-PATH3DU is developed/maintained by SSPA (a third party) and is not directly included in Visual MODFLOW Flex. It is available for free via registered download at the SSPA website for mod-PATH3DU.

Visual MODFLOW Flex currently only supports mod-PATH3DU versions 2.12 and later.

Please Note: MOD-PATH3DU version 2.12 currently only supports fully structured grids or unstructured grids with continuous layers. In other words, the lateral discretization must be the same for all layers in the model. The Visual MODFLOW Flex grid generator for Q-Grids does not generate such grids where pinchouts are specified, therefore the use of V-Grids is recommended for conceptual models with pinchouts when using MODFLOW-USG. Version 3, likely to be released in spring 2019, will support discontinuous layers in unstructured grids.

Working with Particles in Visual MODFLOW Flex

At the Define Particles step, you can create, modify and view groups of Forward and Backward Particles, including those that have been imported from a Visual MODFLOW Classic model.

Assign

You can assign particles using one of the following methods:
Points...

- First, select the model layer, row, or column where you want the circle to be assigned.
- Select the option [Assign > Circle] from the toolbox.
- Using the mouse, left-click on the Layer, Row, or Column view at the desired locations for the particles.
- Once finished, right-click and select "Finish", or click on the Finish button from the toolbox, and the following window will appear:

The following data inputs will be available:

- **Particle group name**: A unique name for the particle group
- **Select Object**: not applicable as you have defined the location of the particles
- **Particle Propagation**: provides the option to select whether the particles will propagate forwards or backwards in time
- **Assign Elevation**: (layer view only)
- **Specified Layers**: the Assign to layer method will be used to assign particles to specified layers as described below
- **Specified Elevation**: the Custom elevations method will be used to assign particles by elevation as described below

**Release Time Tab**: provides the option to select when the particle group is released (i.e. tracking is initiated)
- **Beginning of Simulation**: particles in the group are released at relative model time = 0; this is the default for forwards-in-time particles
- **End of Simulation**: particles in the group are released at the end of the model simulation period; this is the default for backwards-in-time particles
- **Custom**: particles are released at the specified (relative) time based on the time units specified in the project settings.

**End Time Tab**: provides the option to select when particle tracking is terminated (if the particle has not yet reached an endpoint such as a sink for forwards particles and a source for backwards particles). **Note** this applies only to mod-PATH3DU simulations
- **Beginning of Simulation**: remaining particles in the group are terminated at relative model time = 0; this is the default for backwards-in-time particles
- **End of Simulation**: remaining particles in the group are terminated at the end of the model simulation period; this is the default for forwards-in-time particles
- **Custom**: remaining particles in the group are terminated at the specified (relative) time based on the time units specified in the project settings.

**Assign to Layer**: if the Assign Elevation option is specified layers, this option provides you with a table with entries for each layer where you can assign the particles to the specified layer(s) and whether the particles will be assigned to the layer Top, Center (middle), or Bottom.

**Custom elevations**: if the Assign Elevation option is specified elevation, this option provides you with following inputs
- **Method**: Choose from:
  - **Constant**: a specified constant elevation based on the entered Value
  - **Surface**: a surface data object from the project explorer (use the blue arrow to choose an item selected in the Data Explorer)
  - **Horizon**: a horizon object from the model explorer window (use the blue arrow to choose an item selected in the Model Explorer)
  - **Water Table**: a water table object from the model explorer window (use the blue arrow to choose an item selected in the Model Explorer)

- **Click OK when you have finished.**

**Circle...**
A circle of backward tracking particles is typically assigned around a pumping well in order to determine the capture zone for that well.

- **First, select the model layer where you want the circle to be assigned.**
- **Select the option [Assign > Circle] from the toolbox**
- **Move your mouse cursor to the location where you want the center of the circle to be located (typically on top of a well cell, at the center of the cell)**. This option is available in Layer View only
Left-click once to create the circle
The "Define Particles Window" will appear

- Define the Radius for the Circle
- Define the # Particles to be inserted
- Define the Particle Propagation Direction, Assign elevation options, and Release time, as explained above.
- Click OK when you have finished.

Polyl ine...
- First, select the model layer, row, or column where you want the polyline to be assigned.
- Select the option [Assign > Polyline...] from the toolbox.
- Using the mouse, left-click on the Layer, Row, or Column views to draw the polyline in the desired view.
- Once finished, right-click and select “Finish”, or click on the Finish button from the toolbox, and the following window will appear:
The following data inputs will be available in addition to those described above:

- **Particles on polyline:**
  - *Evenly spaced:* the number specified particles will be placed using even spacing along the pathline
  - *Drop particle at every:* particles will be placed along the polyline using the spacing provided, in project length units

- Click OK when you have finished.

**Polygon...**

- First, select the model layer, row, or column where you want the polygon to be assigned.
- Select the option [Assign > Polyline...] from the toolbox.
- Using the mouse, left-click on the Layer, Row, or Column views to draw the polygon in the desired view.
- Once finished, right-click and select "Finish", or click on the Finish button from the toolbox, and the following window will appear:

  ![Assign Polygon in Layer View](image1)
  ![Assign Polygon in Row or Column Views](image2)
The following data inputs will be available in addition to those described above:

- **Particles on polygon**: particles will be placed on a rectangular grid within the polygon using the specified number of columns and rows
- Click OK when you have finished.

**Using Data Object**

- Select the option [Assign > Using Data Object] from the toolbox and the following window will appear:
From the Data Explorer, select a Points, Polyline, or Polygon data object that represents the features/locations where particles are desired. If you do not have a file containing these features, then you can create and digitize new points, polylines, or polygons using the drawing tools. Refer to Creating New Data Objects for more details.

Click the button to insert the data object into the input field. The window will update and provide the same options as assigning by points..., polyline..., or polygon... as described above based on the input type.

**Please Note:** In addition specifying the particle elevations by layers or specified elevations, you may also use the elevations of the feature objects.

Click OK when you have finished.

**Delete Particles**

[Delete >] By Drawing a Box, allows you to delete just a set of particles. Note that this option will only delete the particle type that is active at that time (either Forward or Backward). Before proceeding with the Delete, be sure you have the desired particle type layer as active (and visible in the 2D Viewer). To do this, click on , and move the desired layer type (Forward or Backward particles) to the top of the layer list. Once complete, select Delete > Draw in a Box, and left click with your mouse to anchor the corner of a box around the selected particles you wish to delete, then draw the outline of the box, and release the mouse button to complete the box. The particles that lie within that box will be deleted.
though only for the current model layer.

- [Delete >] Current Layer deletes all particles (Forward and Backward) in the current layer
- [Delete >] All Layers, deletes all particles in all layers

**MODPATH Options**

There are several settings for the MODPATH run, which can be define at the “Translate” step. For more details, see MODPATH

### 10.9 Select Engines

The Single Run workflow step is used to select the Flow and Transport Engines you wish to include in the model run. Visual MODFLOW Flex requires you to select a flow engine for a particular run. Running a transport engine, a particle track engine, and/or ZONEBUDGET is typically optional.

The flow and transport engine(s) available for selection are based on the flow and transport options chosen at the Define Modeling Objectives workflow step as well as the selected grid type. For example, parent/child grids used with MODFLOW-LGR flow models are not currently compatible with any of the transport engines, ZONEBUDGET, MODPATH, or MODPATH3DU. However, model results from an LGR run can be extracted to a standard finite difference subgrid and subsequently run with transport, ZONEBUDGET, MODPATH, or MODPATH3DU.

The table below lists all supported engines and utilities supported in Visual MODFLOW Flex. For more information on which flow and transport engines are compatible with the supported modeling objectives, see the respective sections on Flow and Transport modeling objectives.
Engines supported in the Numerical Workflow

<table>
<thead>
<tr>
<th>Flow</th>
<th>Transport</th>
<th>Utilities</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODFLOW-2000</td>
<td>MT3DMS</td>
<td>ZONEBUDGET and</td>
</tr>
<tr>
<td>MODFLOW-2005</td>
<td>RT3D</td>
<td>MODPATH</td>
</tr>
<tr>
<td>MODFLOW-NWT</td>
<td>MODFLOW-SURFACT</td>
<td>MOD-PATH3DU</td>
</tr>
<tr>
<td>MODFLOW-LGR</td>
<td>MODFLOW-SURFACT</td>
<td></td>
</tr>
<tr>
<td>MODFLOW-SURFACT</td>
<td>SEAWAT</td>
<td></td>
</tr>
<tr>
<td>SEAWAT</td>
<td>SEAWAT</td>
<td></td>
</tr>
</tbody>
</table>

NOTES:

- MODFLOW-LGR should only be run when you have defined a child grid and ran Conceptual to Numerical Conversion from the Conceptual Modeling workflow.
- MT3DMS and MODPATH are not supported by MODFLOW-LGR.
- Transport Engines and MODPATH are not supported by MODFLOW-USG.
- MODFLOW-SURFACT and SEAWAT are coupled flow and transport models and are incompatible with MT3DMS or RT3D.

When you are finished, click (Next Step) button to proceed.

10.10 Translation Settings

The [Translate] button will translate the model input data from Visual MODFLOW Flex file formats to the data files required for the selected numeric engines (see Appendix A).

The available translation settings will depend on what engines you chose to include, in the previous step (Select Run Type / Single Run).
This chapter will cover the following topics:

- **MODFLOW Translation Settings**:
  - General
  - Solvers
  - Recharge and Evapotranspiration
  - Layer Types
  - Cell Rewetting
  - Initial Heads
  - Anisotropy
  - Output Control
  - Advanced Settings

- **MODPATH Translation Settings**

- **MOD-PATH3DU Translation Settings**
  (linked here, but included in the chapter on MODFLOW-USG)

- **MT3DMS Translation Settings**

### 10.10.1 MODFLOW

This chapter describes the features, functionality, and options in the Translation section of the Visual MODFLOW Flex interface. The Translation section is where the MODFLOW packages are generated.
Depending on the numeric engine selected for running the flow simulation, the MODFLOW tree item may appear as MODFLOW-2000, MODFLOW-2005, MODFLOW-NWT, MODFLOW-LGR, MODFLOW-SURFACT, or SEAWAT.

**Please Note:** SEAWAT is a coupled variable-density flow and transport model that is based on linking modified versions of the MODFLOW-2000 and MT3DMS engines. As such, the translation settings related to transport for MODFLOW-2000 also apply to SEAWAT. For more information on the SEAWAT-specific packages see the [SEAWAT](#) section.

### 10.10.1.1 General

Here you can see the output directory where the translated files will be generated.

You can also see the start date. The start date will be the same as that which you specified in the [Define Modeling Objectives](#) step.

### 10.10.1.2 MODFLOW Settings

The following settings are available for the active MODFLOW engine:

**Property Package:** options are available according to which flow engine is active. For:

- **MODFLOW-2000, MODFLOW-2005, MODFLOW-LGR:** the layer-property flow (LPF), block-centered flow (BCF), or hydrogeologic-unit flow (HUF) packages may be used
- **MODFLOW-NWT:** the upstream weighting (UPW) package must be used, and for
- **MODFLOW-SURFACT:** the block-centered flow package (BCF4) must be used.

**Run Type:** Steady-State or Transient

If the Steady-State Flow option is selected, Visual MODFLOW Flex will prepare the data set for a steady-state flow simulation, and will automatically use the data from the first time period (only) of each boundary condition and pumping well defined in Visual MODFLOW Flex to run the model to achieve flow equilibrium (i.e. a time-independent solution since all inputs are constant).
Please Note: if running a steady-state model, at least one boundary condition with a head of known elevation must be specified (e.g. constant head, river stage, drain bottom, general head elevation).

If the Transient Flow option is selected, Visual MODFLOW Flex will prepare the data set for a transient flow simulation. During this process, Visual MODFLOW Flex will automatically merge all of the different time period data defined for each pumping well and boundary condition into the stress period format required by the different versions of MODFLOW. This creates a time-dependent flow solution, as the model is being run with different inputs at different times.

Steady-State Simulation Time: specified time for the mass balance terms in a steady-state run.

Save the .FLO file: toggle to save the .FLO file which can be used for MODPATH and transport (MT3DMS/RT3D) runs.

Stress Periods and Time Steps

The Time Steps option is only available when you are running a transient model (i.e. when Transient Flow run type is selected). For transient flow simulations, Visual MODFLOW Flex will automatically merge all of the different time periods defined for all of the different pumping wells and boundary conditions into the uniform stress period format required by MODFLOW. A stress period is defined as a time period in which all the stresses (boundary conditions, pumping rates, etc.) on the system are constant. Unfortunately, the data collected for each modeling site is rarely synchronized in terms of stress periods, so Visual MODFLOW Flex merges the time schedules for all pumping wells and boundary conditions to determine the length of each stress period for a transient simulation. As a result, the user cannot directly modify the number of stress periods or the length of each stress period.
The Time step options window (as shown in the following figure) is used to define the number of Time steps in each stress period and the time step Multiplier is used to increment each time step size.

The Stress Period column indicates the stress period number while the Start and End columns indicate the start time and end time, respectively, for each stress period.

Each stress period is divided into a user-defined number of Time steps whereby the model will calculate the head solution at each time step. The default value for Time steps is 10.

The time step Multiplier is the factor used to increment the time step size within each stress period (i.e. it is the ratio of the value of each time step to that of the preceding time step). The default value is 1.2. A time step Multiplier value greater than 1 will produce smaller time steps at the beginning of a stress period resulting in a better representation of the changes of the transient flow field. Thus increasing the number of time steps in a simulation may result in smoother head or drawdown versus time curves.

The steady-state column indicates if the stress period is transient or steady-state. This option is available if MODFLOW-2000, MODFLOW-2005, or MODFLOW-SURFACT is selected as the numeric engine for the flow model. These engines allow individual stress periods in a single simulation to be either transient or steady state instead of requiring the entire simulation to be either steady state or transient. Steady-state and transient stress periods can occur in any order. Commonly the first stress period may be run as steady state, to produce a solution that is used as the initial condition for subsequent transient stress periods.

**Accommodating Boundary Conditions with Varying End Times**

Visual MODFLOW Flex allows you to define boundary conditions with varying end times; the number of stress periods for the model will be calculated from the end time of the longest boundary condition. If there is a stress period where there are no data defined for a boundary condition, then you will see an entry of "0" in the appropriate package, for that stress period; this denotes that zero cells are defined for that stress period. In the case of pumping wells, you will see a rate of 0 for the well cells.
10.10.1.3 LGR settings

The Local Grid Refinement (LGR) package, which is only available for MODFLOW-LGR simulations:

1. Identifies the parent and child grids,
2. Specifies how they related to one another, and
3. Controls the coupling iterations between them.

Visual MODFLOW Flex automatically handles the first two items, while the coupling between the parent and child models are defined using the following settings:

- **MXLGRITER**: the maximum number of LGR iterations, a positive integer value
- **RELAXH**: relaxation factor for heads, values must be between 0 and 1
- **RELAXF**: relaxation factor for fluxes, values must be between 0 and 1
- **HCLOSELGR**: head closure criterion for LGR coupling iterations
- **FCLOSELGR**: flux closure criterion for LGR coupling iterations

For mode information please refer to the MODFLOW-LGR Documentation by Mehl and Hill (2013) or the USGS Online Guide.
10.10.1.4 Solvers

Visual MODFLOW Flex comes with a choice of different solvers to use in solving the numerical equations for the flow simulation:

- Preconditioned Conjugate-Gradient Package (PCG2)
- Strongly Implicit Procedure Package (SIP)
- Slice-Successive Overrelaxation Package (SOR)
- WHS Solver for VMOD Flex (WHS)
- Geometric Multigrid Solver (GMG)
- Algebraic Multigrid Methods for Systems (SAMG) and Algebraic Multigrid Solver (AMG) (only available with MODFLOW-2000, 2005 and MODFLOW-LGR)

These solvers and their individual settings can be accessed by selecting MODFLOW/Solver from the Run section of Visual MODFLOW Flex. A Solver Setting window will appear, similar to the image shown in the following figure, with a list for choosing the desired Solver and a listing of the settings for the selected Solver.

Please Note: the SAMG solver is available in the editions of Visual MODFLOW Flex as follows:

- Basic Edition: not available
- Professional Edition: SAMG - single core only
- Premium Edition: SAMG - multi-core/parallel processing

10.10.1.4.1 PCG

The PCG solver (PCG2, in MODFLOW-SURFACT) uses the preconditioned conjugate-gradient method to solve the simultaneous equations produced by the model. Linear and non-linear flow conditions may be simulated. PCG includes two preconditioning options: modified incomplete Cholesky preconditioning, which is efficient on scalar computers; and polynomial preconditioning, which requires less computer storage and, with computer specific modifications, is most efficient on vector computers. Convergence of the solver is determined using both the head-change and residual criteria. Non-linear problems are solved using the
Picard iterations. The PCG Package is described in USGS Water-Resources Investigations Report 90-4048 (Hill, 1997).

The PCG solver works on a two-tier approach to a solution at one time step, inner and the outer iterations. Outer iterations are used to vary the pre-conditioned parameter matrix in an approach toward the solution. An outer iteration is where the hydrogeologic parameters of the flow system are updated (i.e., transmissivity, saturated thickness, storativity) in the preconditioned set of matrices. The inner iterations continue until the user-defined maximum number of inner iterations are executed, or the final convergence criteria are met. The outer iterations continue until the final convergence criteria are met on the first inner iteration after an update.

The following is a description of the solver parameters for the PCG method:

- **Maximum Number of Outer Iterations**: [Default = 25] This parameter provides an upper limit on the number of outer iterations to be performed. The maximum number of iterations will only be used if a convergent solution is not reached beforehand. Twenty-five iterations should be adequate for most problems. However, if the maximum number of outer iterations is reached and an appropriate mass balance error is not achieved, this value should be increased.

- **Maximum Number of Inner Iterations**: [Default = 10] This parameter provides an upper limit on the number of inner iterations to be performed. This number of iterations will only be used if a convergent solution for the current set of matrices in the "outer" iteration is not reached beforehand. Ten inner iterations should be adequate for most problems. More than ten iterations will not usually improve the solution, as the solution is updated again when it returns to the outer iterations.

- **Head Change Criterion for Convergence**: [Default = 0.01] After each outer iteration has completed, the solver checks for the maximum change in the solution at every cell. If the maximum change in the solution is below a set convergence tolerance (set here in the working units feet or meters) then the solution has converged and the solver stops, otherwise a new outer iteration starts.

- **A solution accurate to 0.01 [ft. or m] will normally be sufficient for most problems, unless the maximum head difference across the modeled domain is less than one foot or meter. If an appropriate mass balance is not achieved and the number of inner and outer iterations are within the maximums declared above, this value can be decreased by an order of magnitude, e.g. 0.001.**

- **Residual Criterion for Convergence**: [Default = 0.01] While the head change criterion is used to judge the overall solver convergence, the residual criterion is used to judge the convergence of the inner iterations of the solver. If the maximum absolute value of the residual at all nodes is less than the tolerance specified here (units of L³/T) then the solver will proceed to the next outer iteration.

If you notice that only a few inner iterations are being performed for all outer iterations, and an appropriate mass balance is not achieved, the Residual Criterion value can be decreased by one or more orders of magnitude.

**Note:** The residual criterion is unit-dependent. The default value of 0.01 is often sufficient if your length units are feet or meters and your time units are seconds. If your time units are not
Damping Factor: [Default = 1] This factor allows the user to reduce (dampen) the head change calculated during each successive outer iteration. For most "well posed" and physically realistic groundwater flow problems, the damping factor of one will be appropriate. This parameter can be used to make a non-convergent (oscillating or divergent) solution process more stable such that a solution will be achieved. This is done by decreasing the damping factor to a value between 0 and 1 (only rarely < 0.6). This parameter is similar to the "acceleration parameters" used in other solvers.

Printout Interval: [Default =10] The printout interval is the number of iterations after which the maximum head change (and residual) of the solution is written to the listing (.LST) file.

If the Preconditioning Method is set to Cholesky, the Relaxation parameter can be set. Although the default is 1, in some cases a value of 0.97-0.99 may reduce the number of iterations required for convergence.

10.10.1.4.2 GMG

The GMG solver, based on the preconditioned conjugate gradient algorithm, has been developed by the USGS for solving finite-difference based flow models. As opposed to AMG, the preconditioning in GMG is based on a solver method known as geometric multigrid. The GMG solver has been demonstrated to greatly reduce model run times relative to other solvers using a comparable amount of memory. Detailed information about the GMG solver, including comparisons with the AMG solver, can be found in the GMG Linear Equation Solver Package PDF documentation.

The solver parameters for the Geometric Multigrid Solver are described below using excerpts from the GMG Linear Equation Solver Package PDF documentation:

- Max. outer iterations (MXITER): The maximum number of outer iterations. For linear problems, MXITER can be set to 1. For nonlinear problems, MXITER needs to be larger, but rarely more than 100. The maximum number of iterations will only be used if a convergent solution is not reached beforehand.

- Max. inner iterations (IITER): The maximum number of PCG iterations for each linear solution. A value of 100 is typically sufficient. It is frequently useful to specify a smaller number for nonlinear problems so as to prevent an excessive number of inner iterations. This number of iterations will only be used if a convergent solution for the current set of matrices in the "outer" iteration is not reached beforehand.

- Adaptive Damping Control (IADAMP): IADAMP is a flag that controls adaptive damping. If IADAMP = 0, then the value assigned to DAMP is used as a constant damping parameter. If IADAMP ≠ 0, then the value of DAMP is used for the first nonlinear iteration. The damping parameter is adaptively varied on the basis of the head change, using Cooley's method for subsequent iterations.

- Head change criterion (HCLOSE): After every outer iteration is completed, the solver checks for the maximum change in the solution at every cell. If the maximum change in
the solution is below a set convergence tolerance (set here in the working units of feet or metres) then the solution has converged and the solver stops, otherwise a new outer iteration is started. A solution accurate to 0.01 [ft. or m] will normally be sufficient for most problems unless the maximum head change throughout the modeled domain is less than 1 foot or metre. If an appropriate mass balance is not achieved and the number of inner and outer iterations is within the maximums, this value can be decreased by an order of magnitude.

- **Residual criterion (RCLOSE):** RCLOSE is the residual convergence criterion for the inner iteration. The PCG algorithm computes the l2-norm of the residual and compares it against RCLOSE. Typically, RCLOSE is set to the same value as HCLOSE (see below). If RCLOSE is set too high, then additional outer iterations may be required due to the linear equation not being solved with sufficient accuracy. On the other hand, a too restrictive setting for RCLOSE for nonlinear problems may force an unnecessarily accurate linear solution. This may be alleviated with the IITER parameter or with damping.

- **Relaxation parameter (RELAX):** The RELAX parameter can be used to improve the spectral condition number of the ILU preconditioned system. The value of RELAX should be approximately one. However, the relaxation parameter can cause the factorization to break down. If this happens, then the GMG solver will report an assembly error and a value smaller than one for RELAX should be tried. This item is read only if ISC = 4.

- **Upper bound of estimate (NPBOL):** IOUTGMG is a flag that controls the output of the GMG solver. The possible values of IOUTGMG and their meanings are as follows: If IOUTGMG = 0, then only the solver inputs are printed. If IOUTGMG = 1, then for each linear solve, the number of PCG iterations, the value of the damping parameter, the l2-norm of the residual, and the max-norm of the head change and its location (column, row, layer) are printed. At the end of a time/stress period, the total number of GMG calls, PCG iterations, and a running total of PCG iterations for all time/stress periods are printed. If IOUTGMG = 2, then the convergence history of the PCG iteration is printed, showing the l2-norm of the residual and the convergence factor for each iteration. IOUTGMG = 3 is the same as IOUTGMG = 1 except output is sent to the terminal instead of the MF2K LIST output file. IOUTGMG = 4 is the same as IOUTGMG = 2 except output is sent to the terminal instead of the MF2K LIST output file.

- **Multigrid Preconditioner (ISM):** ISM is a flag that controls the type of smoother used in the multigrid preconditioner. The possible values for ISM and their meanings are as follows: If ISM = 0, then ILU(0) smoothing is implemented in the multigrid preconditioner. This smoothing requires an additional vector on each multigrid level to store the pivots in the ILU factorization. If ISM = 1, then Symmetric Gauss Seidel (SGS) smoothing is implemented in the multigrid preconditioner. No additional storage is required for this smoother; users may want to use this option if available memory is exceeded or nearly exceeded when using ISM = 0. Using SGS smoothing is not as robust as ILU smoothing; additional iterations are likely to be required in reducing the residuals. In extreme cases, the solver may fail to converge as the residuals cannot be reduced sufficiently.

- **Semoaarsening Control in the Multigrid Preconditioner (ISC):** A flag that controls semicoarsening in the multigrid preconditioner. The possible values of ISC and their meanings are given as follows: If ISC = 0, then the rows, columns and layers are all coarsened. If ISC = 1, then the rows and columns are coarsened, but the layers are not. If ISC = 2, then the columns and layers are coarsened, but the rows are not. If ISC
If ISC = 3, then the rows and layers are coarsened, but the columns are not. If ISC = 4, then there is no coarsening. Typically, the value of ISC should be 0 or 1. In the case that there are large vertical variations in the hydraulic conductivities, then a value of 1 should be used. If no coarsening is implemented (ISC = 4), then the GMG solver is comparable to the PCG2 ILU(0) solver described in Hill (1990) and uses the least amount of memory.

- Damping factor (DAMP): This factor allows the user to reduce (dampen) the head change calculated during each successive outer iteration. For most "well posed" and physically realistic groundwater flow problems, the dampening factor of one will be appropriate. This parameter can be used to make a non-convergent (oscillating or divergent) solution process more stable such that a solution will be achieved. This is done by decreasing the damping factor to a value between 0 and 1 (only rarely < 0.6). This parameter is similar to "acceleration parameters" used in other solvers.

10.10.1.4.3 WHS

The WHS Solver uses a Bi-Conjugate Gradient Stabilized (Bi-CGSTAB) acceleration routine implemented with Stone incomplete decomposition for preconditioning of the groundwater flow partial differential equations. This solver, as all iterative solvers, approaches the solution of a large set of partial differential equations iteratively through an approximate solution. Because the matrix equation for groundwater flow is initially "ill-conditioned", effective preconditioning of these matrices is necessary for an efficient solution.

The WHS solver works on a two-tier approach to a solution at one time step. Outer iterations are used to vary the factorized parameter matrix in an approach toward the solution. An outer iteration is where the hydrogeologic parameters of the flow system are updated (i.e., transmissivity, saturated thickness, storativity) in the factorized set of matrices. Different levels of factorization allow these matrices to be initialized differently to increase the efficiency of solution and model stability. Inner iterations are used to iteratively solve the matrices created in the outer iterations.

The solver parameters for the WHS method are described below:

- Maximum Number of Outer (non-linear) Iterations: [Default = 50] This parameter provides an upper limit on the number of outer iterations to be performed. The maximum number of iterations will only be used if a convergent solution is not reached beforehand. Fifty iterations should be adequate for most problems. However, if the maximum number of outer iterations is reached and an appropriate mass balance error is not achieved, this value should be increased.
- Maximum Number of Inner Iterations: [Default = 25] This parameter provides an upper limit on the number of inner iterations to be performed. This number of iterations will only be used if a convergent solution for the current set of matrices in the "outer" iteration is not reached beforehand. Twenty-five inner iterations should be adequate for most problems. However, if the maximum number of inner iterations was used for all outer iterations and an appropriate mass balance error was not achieved, this value can be increased.
- Head Change Criterion for Convergence: [Default = 0.01] After every outer iteration is completed, the solver checks for the maximum change in the solution at every cell. If the maximum change in the solution is below a set convergence tolerance (set here in
the working units of feet or metres) then the solution has converged and the solver stops, otherwise a new outer iteration is started. A solution accurate to 0.01 [ft. or m] will normally be sufficient for most problems unless the maximum head change throughout the modeled domain is less than 1 foot or metre. If an appropriate mass balance is not achieved and the number of inner and outer iterations is within the maximums, this value can be decreased by an order of magnitude.

- **Residual Criterion for Convergence:** [Default = 0.01] While the head change criterion is used to judge the overall solver convergence, the residual criterion is used to judge the convergence of the inner iterations of the solver. If the change in successive inner iterations is less than the tolerance specified here (in working units of feet or metres), then the solver will proceed with the next outer iteration. The residual criterion for convergence of 0.001 should be appropriate for most problems. However, if you notice that only a few inner iterations are being performed for every outer iteration and an appropriate mass balance is not achieved, this parameter value can be decreased by one or more orders of magnitude.

- **Damping Factor for the Outer Iterations:** [Default = 1] This factor allows the user to reduce (dampen) the head change calculated during each successive outer iteration. For most "well posed" and physically realistic groundwater flow problems, the dampening factor of one will be appropriate. This parameter can be used to make a non-convergent (oscillating or divergent) solution process more stable such that a solution will be achieved. This is done by decreasing the damping factor to a value between 0 and 1 (only rarely < 0.6). This parameter is similar to "acceleration parameters" used in other solvers.

- **Relative Residual Criterion:** [Default = 0] This parameter provides another method of checking for convergence of the inner iteration. This method compares the residual from the most recent inner iteration to the residual from the initial inner iteration. Once the most recent inner iteration residual is below the initial inner iteration residual times the relative residual criterion, the current outer iteration is completed and a new outer iteration will be started.

- **Factorization Level:** [Default = 0] There are two “levels” of factorization available with the WHS solver, 0 and 1. Level 0 requires more outer iterations but less memory. Level 1 requires fewer outer iterations but more memory. While convergence of the solver requires fewer iterations with a factorization level of 1, the memory required to run the solver increases with this factorization level. Also, the work per iteration increases with the level 1 factorization such that the total solution time may not be less than the solution time using level 0 factorization.

10.10.1.4.4 SIP

The Strongly Implicit Procedure, also known as SIP, is a method for solving a large system of simultaneous linear equations by iterations. The advantage of the SIP solver is that it is very stable and generally converges to a solution, but often very slowly. It is not as fast as the PCG method, but it requires less memory to compute the final solution. Because each equation involves up to seven unknown values of head, and because the set of unknown values changes from one equation to the next throughout the grid, the equations for the entire grid must be solved simultaneously at each time step. This package is described in Chapter 12 of the MODFLOW manual included with your Visual MODFLOW Flex media, in the Manual folder.
The solver parameters for the SIP method are described below:

- **Maximum Number of Iterations**: [Default = 200] This is the upper limit on the number of iterations to be performed. The maximum number of iterations will only be considered if a convergent solution is not reached beforehand. Two hundred iterations should be adequate for most problems. However, if the maximum number of iterations is reached and an appropriate mass balance error is not achieved, this value should be increased.

- **Number of Iteration Parameters**: [Default = 5] The finite difference equations describing the groundwater flow system can be put into matrix form as \([A] \{h\} = \{q\}\). Where \([A]\) is the coefficient matrix, \(\{h\}\) is the heads array and \(\{q\}\) is the flux array. The number of iteration parameters indicates the number of parameters that will be used to transform the initial coefficient matrix \([A]\) to a similar matrix that can be decomposed into two lower and upper triangular matrices \([L]\) and \([U]\), respectively. The default value of 5 is generally sufficient.

- **Acceleration Factor**: [Default = 1] The acceleration factor controls the magnitude of head change between iterations. The acceleration factor must be positive. Values larger than one will result in larger head changes between iterations; the solution may be approached faster but it may also overshoot the solution more easily. Values less than one will result in smaller head changes, requiring more iterations to reach a solution.

- **Head Change Criterion for Convergence**: [Default = 0.01] After each iteration is completed, the solver checks for the maximum change in the solution at every cell. If the maximum change in the solution is below a set convergence tolerance (set here in the working units of feet or metres) then the solution has converged and the solver stops, otherwise a new iteration is started. A solution accurate to 0.01 [ft. or m] will normally be sufficient for most problems unless the maximum head change throughout the modeled domain is smaller than one foot or metre. If an appropriate mass balance is not achieved and the maximum number of iterations is not reached, this value can be decreased by an order of magnitude.

- **Printout Interval**: [Default = 10] The printout interval is the number of iterations after which the maximum head change (and residual) of the solution is written to the listing (.LST) file.

- **User Seed Value**: [Default = 0.01] There are two options: either the user can enter the seed, or the seed will be calculated at the start of the simulation from problem parameters. The iteration parameter ‘seed’ is used as a basis for determining the sequence of \(w\) values. The \(w\) multiplies each term on the right side of the equation; and must be cycled through a series of values in successive iterations to achieve satisfactory rates of convergence. The more strongly diagonal the coefficient matrix, the less important the choice of seed will be.

---

10.10.1.4.5 SOR

Slice-Successive Over-Relaxation is a method for solving large systems of linear equations iteratively. It is implemented in the SOR Package by dividing the finite difference grid into vertical slices, and grouping the node equations into discrete sets, each set corresponding to a slice. In every iteration, these sets of equations are processed in turn, resulting in a new set of estimated head values for each slice. As the equations for each slice are processed, they are first expressed in terms of the changes in computed heads between successive iterations. The set of equations corresponding to the slice is then solved directly by Gaussian elimination, treating the terms for adjacent slices as known quantities. The values of head change computed for the slice are then each multiplied by an acceleration variable, \(T\). The
results are taken as the final values of head change in that iteration for the slice. This procedure is repeated for each slice in sequence until all of the slices in the three-dimensional array have been processed, thus completing a domain iteration. The entire sequence is then repeated, until the differences between the head values computed in successive iterations are less than the chosen criterion at all nodes in the mesh. The SOR Package is described in detail in Chapter 13 of the MODFLOW reference manual included with your VMOD Flex media, in the Manual folder.

The solver parameters for the SOR method are described below:

- **Maximum Number of Iterations:** [Default = 50] This parameter provides an upper limit on the number of iterations to be performed. The maximum number of iterations will only be used if a convergent solution is not reached beforehand. 50 iterations should be adequate for most problems. However, if the maximum number of outer iterations is reached and an appropriate mass balance error is not achieved, this value should be increased.

- **Acceleration Factor:** [Default = 1] The acceleration factor controls the magnitude of head changes between iterations. The acceleration factor must be positive. Values larger than one will result in larger head changes between iterations; the solution may be approached faster but it may also overshoot the solution more easily. Values less than one will result in smaller head changes, thus, requiring more iterations to reach a solution.

- **Head Change Criterion for Convergence:** [Default = 0.01] After each iteration is completed, the solver checks for the maximum change in the solution at every cell. If the maximum change in the solution is below a set convergence tolerance (set here in the working units of feet or metres), then the solution has converged and the solver stops, otherwise a new iteration is started. A solution accurate to 0.01 [ft. or m] will normally be sufficient for most problems unless the maximum head change throughout the model domain is less than 1 foot or metre. If an appropriate mass balance is not achieved and the number of iterations is less than the maximum, this value can be decreased by an order of magnitude.

- **Printout Interval:** [Default = 10] The printout interval is the number of iterations after which the maximum head change (and residual) of the solution is written to the listing (.LST) file.

---

Visual MODFLOW Flex supports the Algebraic MultiGrid Methods for Systems Solver (SAMG) Package developed by the Fraunhofer Institute for Algorithms and Scientific Computing (FhG-SCAI).

**Please Note:** the SAMG solver is only compatible with the MODFLOW-2000, -2005 and -LGR flow engines. The SAMG solver is available in single-core mode in Visual MODFLOW PRO and in multi-core mode in Visual MODFLOW Premium.

The Algebraic MultiGrid (AMG) Package solver may be obtained from the Fraunhofer Institute for Algorithms and Scientific Computing (FhG-SCAI) for research purposes only.
The SAMG solver package is a complete multi-level framework, designed to overcome the high memory requirements of previous AMG-based solvers, while maintaining the scalability and rapid execution times. Testing of the SAMG solver vs. the PCG2 solver using several models generated using Visual MODFLOW demonstrated solution times to be faster by a factor of between 2.4 and 11.3. The SAMG Package has some distinct advantages over other solvers available with MODFLOW for problems with large grids (more than about 40,000 cells) and/or a highly variable hydraulic-conductivity field. The advantages of multigrid methods over the other iterative solvers mentioned are (1) the effectiveness of the multigrid solver is not dependent on the initial head distribution, and (2) for many problems of interest, the rate of convergence scales approximately linearly with the size of the domain, unlike the other solvers where the rate of convergence increases nonlinearly (Demmel, 1997).

The Solver settings window contains a number of user-defined solver settings which can influence the speed and effectiveness of the AMG solver.

- **Max. Iterations (MXITER):** [Default = 50] MXITER is the maximum number of times that the AMG routines will be called to obtain a solution. MXITER is never less than 2, and rarely more than 50. MXITER often equals 2 when the problem is linear (all layers are confined, and no boundary conditions are nonlinear; the Evapotranspiration, Drain, and River Packages, for example, produce nonlinear boundary conditions). For nonlinear problems, MXITER generally is 50 or less; however values near 50 and sometimes even larger are needed for more severely nonlinear problems.

- **Max. Cycles (MXCYC):** [Default = 50] For each call to the solver, AMG cycles through one or more sequences of coarsening and refinement. The solver is limited to a maximum of MXCYC cycles per call to the solver. For most problems, convergence for each iteration is achieved in less than 50 cycles, so that generally MXCYC can be less than 50. For highly nonlinear problems, however, better performance may be achieved by limiting the solver to a small number of cycles, and increasing the maximum number of iterations (MXITER). This prevents the solver from needlessly finding very accurate solutions at early iterations of these highly nonlinear problems.

- **Residual Convergence Criterion (RCLOSE) for the inner iteration.** Typically RCLOSE is set to the same value as HCLOSE. If RCLOSE is set too high, then additional outer iterations may be required due to the linear equation not being solved with sufficient accuracy. Likewise, a too restrictive setting for RCLOSE for nonlinear problems may force an unnecessarily accurate linear solution. This may be alleviated with the MXCYC parameter or with damping.

- **Note:** In the new SAMG package, RCLOSE and HCLOSE replace BCLOSE.

- **Damping Factor (DAMP):** [Default = 1] The damping factor can be used to restrict the head change from one iteration to the next, which commonly is useful in very nonlinear problems. DAMP makes the solution change slowly, thus avoiding spurious deviations prompted by nonlinear effects at intermediate solutions. Values of DAMP less than 1.0 restrict the head change (under-relaxation), while values greater than 1.0 accelerate the head change (over-relaxation). For linear problems, no damping is necessary, and DAMP should be set equal to 1.0. For non-linear problems, restricting the head change (DAMP < 1.0) may be necessary to achieve convergence, and values of DAMP between 0.5 and 1.0 are generally sufficient.

For some nonlinear problems, imposing a fixed value of DAMP for every iteration can hinder convergence. One remedy for this condition is to adjust the amount of damping depending on
how the head solution progresses. The AMG Package provides two adaptive damping strategies: (1) Cooley’s method with Huyakorn’s modification, and (2) the relative reduced residual method. These methods are described in detail in the U.S. Geological Open-File Report 01-177. A DAMP value of -1 will utilize the first method, and a DAMP value of -2 will utilize the second method.

- Max. Damping Factor (DUP): [Default = 1] The upper limit for DAMP when an adaptive damping strategy is used.
- Min. Damping Factor (DLOW): [Default = 0.2] The lower limit for DAMP when an adaptive damping strategy is used
- Head Change Convergence Criterion (HCLOSE), similar as described for previous solvers
- Perform Conjugate Gradient Iterations (ICG): [Default = checked] In some cases, AMG can perform poorly as a result of a small number of error components that are not reduced during the AMG cycling. A few iterations of a conjugate gradient solver can often reduce these error components and thus help convergence (Cleary and others, 2000). In these cases, the parameter ICG can be set to 1 to perform conjugate gradient iterations at the end of each multigrid cycle. Activating this option can decrease execution times for some problems, but it will also increase the amount of memory used by the solver.
- The Print Flag (IOUTAMG) frame allows you to select between various print options.
- CONTROL Parameter [Default = 2]
  - 1 - reuse of the setup phase is not used
  - 2 - reuse of the setup phase will be used (Recommended)
  - 3 - reuse of the setup phase will be used, and SSC will be used

The NWT Solver parameters are shown in the image below. Detailed information about MODFLOW-NWT can be found:
http://water.usgs.gov/ogw/modflow-nwt/

**NOTE:** MODFLOW-NWT includes an upstream-weighting (UPW) intercell conductance package as a replacement internal flow package to those provided by the BCF, LPF, and HUF Packages. When MODFLOW-NWT is selected as the flow engine, the UPW property package will be generated during translation and used in the model run.
Visual MODFLOW Flex provides options for the most-commonly used parameters (head change criterion, flux change criterion, number of iterations, etc.) and to choose the solver OPTIONS (Automate Solver Options). MODFLOW-NWT provides flexibility to specify the values for each of the parameters, or to choose from one of the pre-defined solver “schemes”. When you select one of the pre-defined schemes, then the solver parameters are fixed and cannot be modified (all fields will be non-visible). If choose the "SPECIFIED" option, then the full set of solver parameters will be defined in the .NWT file (using defaults) and you will have the ability to change these parameters in the .NWT file after Translation. When you choose one of the pre-defined schemes (Simple, Moderate, or Complex) then you will see only an abbreviated set of parameters.

- **SPECIFIED** indicates that the optional solver input values listed for items 1 and 2 will be specified in the NWT input file by the user.
- **SIMPLE** indicates that default solver input values will be defined that work well for nearly linear models. This would be used for models that do not include nonlinear stress packages, and models that are either confined or consist of a single unconfined layer that is thick enough to contain the water table within a single layer. (See Table 2 of package documentation for the solver input values that will be used for this option.)
- **MODERATE** indicates that default solver input values will be defined that work well for moderately nonlinear models. This would be used for models that include nonlinear stress packages, and models that consist of one or more unconfined layers. The “MODERATE” option should be used when the “SIMPLE” option does not result in successful convergence. (See Table 2 of package documentation for the solver input values that will be used for this option.)
- **COMPLEX** indicates that default solver input values will be defined that work well for highly nonlinear models. This would be used for models that include nonlinear stress packages, and models that consist of one or more unconfined layers representing complex geology and surface water/groundwater interaction. The “COMPLEX” option should be used when the “MODERATE” option does not result in successful convergence. (See Table 2 of package documentation for the solver input values that will be used for this option.)
HEADTOL (units of length)—is the maximum head change between outer iterations for solution of the nonlinear problem.

FLUXTOL (units of length cubed per time)—is the maximum root-mean-squared flux difference between outer iterations for solution of the nonlinear problem.

MAXITEROUT is the maximum number of iterations to be allowed for solution of the outer (nonlinear) problem.

THICKFACT is the portion of the cell thickness (length) used for smoothly adjusting storage and conductance coefficients to zero.

LINMETH is a flag that determines which matrix solver will be used.

- 1 = GMRES will be used
- 2 = xMD will be used

IBOTAV is a flag that indicates whether corrections will be made to groundwater head relative to the cell-bottom altitude if the cell is surrounded by dewatered cells. Checked = a correction will be made; Not-checked = no correction will be made. This input variable is problem specific and both options (IBOTAV selected/not-selected) should be tested.

**NOTE**: The modification to the .WEL package for variable PHIRAMP is currently not supported in Visual MODFLOW Flex

### 10.10.1.5 Recharge and Evapotranspiration

![Recharge options window](image)

**Recharge**

The Recharge zone distribution can be applied to any of the user-specified model Layers. If the recharge is assigned to the top grid layer, and some cells in the top layer become dry during the course of the simulation, or if some cells in the top layer are designated as no-flow cells, the MODFLOW program allows the recharge to be applied to the grid cells in the upper most active (wet) layer in the model. The Recharge settings are shown in the following Recharge options window and these are described below.
• Recharge is only applied to the top grid layer: If any grid cells in Layer 1 are dry, or if they are designated as no-flow cells, the recharge values assigned to these grid cells will NOT be carried down to the underlying active (wet) grid cells. In this case, the inactive or dry cells act like an impermeable barrier to the recharge.
• Recharge is applied to the specified layer: It allows the user to assign the recharge values to any of the specified model layers.
• Recharge is applied to the uppermost active layer: If any grid cells in Layer 1 are dry, or if they are designated as no-flow cells, the recharge values assigned to these grid cells will be carried down to the upper most active (wet) grid cell in the same vertical column of grid cells.

**Note:** A constant head boundary condition always intercepts recharge and prevents deeper infiltration.

---

**Evapotranspiration**

The Evapotranspiration distribution can be applied to any of the user-specified model Layers. If assigned to the top grid layer, and some cells in the top layer become dry during the course of the simulation, or if some cells in the top layer are designated as no-flow cells, the MODFLOW program allows the Evapotranspiration to be applied to the grid cells in the uppermost active (wet) layer in the model. The Evapotranspiration settings are shown in the following Evapotranspiration Options window and these are described below.

• Evapotranspiration is only applied to the top grid layer: If any grid cells in Layer 1 are dry, or if they are designated as no-flow cells, the Evapotranspiration values assigned to these grid cells will NOT be carried down to the underlying active (wet) grid cells. In this case, the inactive or dry cells act like an impermeable barrier to the Evapotranspiration.
• Evapotranspiration is applied to the specified layer: It allows the user to assign the Evapotranspiration values to any of the specified model layers.
• Evapotranspiration is applied to the uppermost active layer: If any grid cells in Layer 1 are dry, or if they are designated as no-flow cells, the evapotranspiration values assigned to these grid cells will be carried down to the upper most active (wet) grid cell in the same vertical column of grid cells.

---

**10.10.1.6 Lakes**

The following settings are supported in Visual MODFLOW Flex for the LAK package and facilitate the internal flow balance for the LAK package:
THETA: variable to specify an explicit (THETA=0.0), semi-implicit (0.0<THETA<1.0), or fully implicit (THETA=1.0) solution for lake stages

MSSITR: Maximum number of iterations for steady-state solution

SSCNCR: Convergence criterion for equilibrium lake stage solution by Newton's method.

**Steady-State Lakes**

The lake package has a known issue for *steady-state simulations* - the initial lake level specific is held constant for the simulation period and will not reach equilibrium with the heads in the surrounding hydrogeologic unit(s). The recommended action to determine the equilibrium lake-level is to simulate a transient run with sufficient duration to estimate the steady-state levels as input to a steady-state simulation.
10.10.1.7 Layer Types

The Layer Type Settings window is used to set the LAYCON value and the LAYAVG variables required by the MODFLOW numeric engine.

The LAYCON value is the layer-type index array recognized by MODFLOW. MODFLOW has four different Layer Types to choose for LAYCON values as described below:

- **Type 0 - Confined:** Transmissivity and storage coefficients of the layer are constant for the entire simulation.
- **Type 1 - Unconfined:** Transmissivity of the layer varies and is calculated from the saturated thickness and hydraulic conductivity. The storage coefficient is constant; valid only for Layer 1.
- **Type 2 - Confined/Unconfined:** Transmissivity of the layer is constant. The storage coefficient may alternate between confined and unconfined values.
- **Type 3 - Confined/Unconfined:** [Default setting] Transmissivity of the layer varies. It is calculated from the saturated thickness and hydraulic conductivity. The storage coefficient may alternate between confined and unconfined values. Vertical leakage from above is limited if the aquifer becomes desaturated.

The LAYAVG value determines the method of computing interblock transmissivity.

Following are the five methods used in assigning the LAYAVG value.

- **00 - Harmonic mean interblock transmissivity** [Default setting for MODFLOW-96, MODFLOW-2000, and MODFLOW-2005].
- **10 - Arithmetic mean interblock transmissivity.**
- **20 - Logarithmic mean interblock transmissivity.**
- **30 - Arithmetic mean saturated thickness times logarithmic mean hydraulic conductivity.**
- **40 - Harmonic mean interblock hydraulic conductivity introduced in BCF4 package** [Default (Required) setting for MODFLOW-SURFACT].
- **50 - Arithmetic mean interblock hydraulic conductivity introduced in BCF4 package.**
Note that the LAYAVG values are two digits with a factor of ten. For example, a LAYCON value of 21 represents an unconfined layer where the interblock transmissivity is calculated using a logarithmic mean.

The Layer column in the Layer Settings window (see following figure) is the layer number which is automatically numbered as one row for each layer of the model grid.

The LAYCON column is the Input LAYCON value, which includes the first digit (tens) stored as the LAYAVG value (Interblock transmissivity), and the second digit (ones) stored as the LAYCON value (Layer type). Thus the one Input LAYCON value holds the identification for each layer of the model grid.

The Interblock transmissivity column displays the LAYAVG value and descriptive name associated with each layer of the model. The available LAYAVG settings can be chosen from a picklist by clicking the down arrow key, or you can scroll through the options by clicking the spin buttons on the left (as shown in the following figure).

The Layer type column displays the Layer Type associated with each layer of the model. The available layer types can be chosen from a picklist by clicking the down arrow on the right, or you can scroll through the options by clicking the spin buttons on the left.

### 10.10.1.8 Cell Rewetting

The original USGS MODFLOW did not allow cells in unconfined layers to become re-saturated if the head dropped below the bottom elevation of the grid cell during the course of the simulation or during the solution iterations. Instead, these cells were simply made inactive for the remainder of the simulation. However, the USGS later extended the Block-Centered-Flow package (BCF2) to allow for the rewetting of these “dry” cells during a transient simulation. While this represented a major advancement for more accurate representations of water table aquifers, it also causes the solution to be much more unstable in some situations. More detailed information on the cell wetting and the BCF package can be found in the MODFLOW Packages Reference Manual included with your Visual MODFLOW media, in the Manual folder.
The Re-wetting settings may be accessed by selecting MODFLOW/Re-wetting from the top menu bar of the Run section. A Dry Cell Wetting Options window will appear as shown in the following figure, and described below.

- **Activate cell wetting (IWDFLG)** is used to indicate if the wetting capability is active (IWDFLG = 1) or inactive (IWDFLG = 0).
- **Wetting threshold** is used to determine if the dry cell needs to be wetted. For a dry cell to become wet, the head in the adjacent grid cell(s) must be greater than the elevation of the bottom of the dry cell plus the Wetting threshold value.
- **Wetting interval (IWETIT)** indicates how often MODFLOW attempts to wet the dry cells detected during the course of the solution iterations. For example, if IWETIT = 2, cell wetting would be attempted at every second iteration. If a model with dry cells is having problems converging to a solution, it may be a result of oscillations due to drying and re-wetting of grid cells in sensitive regions of the model. Setting a larger cell wetting interval value may help the solution to converge, by allowing the heads in neighboring cells to get closer to a better solution before wetting the dry cells.

**Wetting Methods**

The wetting of a dry cell is triggered by the head values in adjacent grid cells (see Wetting threshold above). The Wetting method frame has two options for determining the adjacent grid cells to use for determining whether the dry cell should be wetted.

- **From below (WETDRY < 0)** will use only the head in the grid cell directly below the dry cell to determine if the dry cell should be wetted. This option is generally more stable, and is particularly good when the adjacent horizontal cells are poor indicators of whether a cell should become wet (e.g. areas with steep vertical gradients and variable bottom elevations).
- **Wet cells from side and below (WETDRY > 0)** will use the head in all four adjacent grid cells and the grid cell directly below the dry cell to determine if the dry cell should be wetted. This option is useful in situations where a dry cell is located above a no-flow cell, such that it cannot be re-wet from below. It is also applicable for one-layer models where there are no cells below. However, this method can cause some problems in multi-layer models where inactive cells occur beneath wet cells.

**Wetting Head**

When a dry cell is wetted, the new head may be calculated using one of two methods:

- **Calculated from neighbors**
  \[ \text{Head} = Z_{\text{bot}} + \text{Wetting factor} \times (\text{Neighboring head} - Z_{\text{bot}}) \]

- **Calculated from threshold**
  \[ \text{Head} = Z_{\text{bot}} + \text{Wetting factor} \times \text{Wetting threshold} \]

Where \( Z_{\text{bot}} \) is the elevation of the bottom of the current cell.

Generally, the first equation is thought to be more reasonable, since the cell's new head is varied according to the head in the neighboring cell, which caused it to become wet. However, in situations where MODFLOW is over-estimating head changes during iterations, this
equation may cause non-convergence. The second equation can then be used to attempt a more stable solution.

**Setting Head Values in Dry Cells**

Each dry cell is assigned a default head value as a flag to indicate it is dry. This value is typically a very large negative number (e.g. -1.0e30). However, the presence of large negative head values in dry cells may cause problems for parameter estimation simulations because this large negative value may be used to calculate the calibration residual (calculated head - observed head) at a grid cell that has become dry during one of the PEST iterations. In this case, it is more appropriate to assign the head value in dry cells equal to the cell bottom elevation to avoid this problem.

**Setting Minimum Saturated Thickness**

When PEST runs with varied model parameters, some of these runs may produce dry cells, and as a result MODFLOW assigns head values equal to -1.0e30 to all the dry cells. This can cause the objective function to be skewed, and the subsequent model runs to fail to converge. If the “Keep minimum saturated thickness for the bottom layer” option is activated by assigning an appropriate head value to the bottom cell, MODFLOW will keep the bottom cell saturated. It prevents the column from drying out, ensures that PEST will continue running even though the calculated head is actually below the bottom layer of the model, and helps with model convergence. It is recommended to not activate this option in the first run of a model because it is important to know if the dry cells exist or not, and by preventing the column from drying out, the model results could be misleading.

**Hints For Using Dry Cell Wetting**

Cell re-wetting often promotes a non-converging or unstable solution, which may be indicated by cells cycling between wet and dry. If this happens, we recommend you try the following:

- Set any cells that you know should never become wet to inactive.
- Increase the Wetting threshold value. This makes it more difficult for a cell to be wetted, and therefore helps stop MODFLOW from repeatedly turning a cell on and then off again. However, the solution may become less accurate since cells that should become wet might stay dry.
- Modify the Wetting factor value. This will increase or decrease the new head in cells which are wetted.
- Change the Wetting method that controls cell wetting.
- Change the Wetting head option that calculates the new head in the wetted cell.
- Try using the SIP or PCG solver, and modifying the solver parameters.
- For steady-state solutions, start with good initial head estimates. This will provide good indications of which cells should be wet and dry, and therefore conversions of cells between wet and dry will be minimized.
10.10.1.9 Initial Heads

In Visual MODFLOW Flex, the Initial Heads are defined at the stage of Defining the Properties. For more details, please see the section on setting **Initial Heads** in the Define Properties workflow step.

When translating your model you have the following settings for Initial Head:

- **Use Specified Heads** - will use the values that are defined for “Initial Heads” at the Define Properties step.
- **Use Ground Elevation** - will use the elevation values for the top of Layer 1.
- **Use Previous MODFLOW Run** - requires you to select a .HDS file from a previous MODFLOW Run and specify the Time step to be used.

**Warning!**

Using Heads from Previous MODFLOW Run

The selected .HDS file cannot be the same as the .HDS file in the current translation directory. If you select Use .HDS from Previous MODFLOW Run option, you must choose a .HDS file from another directory.
10.10.1.10 Anisotropy

Horizontal anisotropy is the ratio of transmissivity or hydraulic conductivity along a column to its component value along each row. The Anisotropy settings in Visual MODFLOW Flex may be accessed in the Run section by selecting MODFLOW/Anisotropy from the top menu bar. The Anisotropy Factor window (as shown in the figure below) will provide two choices for determining how the anisotropy is calculated for the model.

- Anisotropy by layer
- Anisotropy as specified

The Anisotropy by layer option uses the anisotropy ratio (Ky/Kx Ratio) specified for each layer of the model (as shown in the figure), and the Kx values specified in each grid cell, to calculate the Kx or Ky values for each grid cell. Selecting Anisotropy by layer will not replace the original modeled values for Ky, but will instead calculate values during the translation of Visual MODFLOW Flex file formats to MODFLOW input data file formats to be used for the run. By default each new simulation will be set to use the Anisotropy by layer option.

The Anisotropy as specified option will use the Kx and Ky values defined for each property zone. This feature allows spatially variable anisotropy within a layer, as opposed to the Anisotropy by layer option which applies a single anisotropy ratio (Ky/Kx Factor) for the entire layer. A more in-depth discussion of spatially variable anisotropy can be found in Kladias (1997).

Note: MODFLOW-2000 does not support running the BCF package with Anisotropy as Specified.
10.10.1.11 Output Control

The Output Control run options set the information and frequency of information written and saved to the various MODFLOW output files (see following figure).
Each MODFLOW simulation can produce three binary output files and one ASCII output file:

- Binary head file (modelname.HDS)
- Binary drawdown file (modelname.DDN)
- Binary flow file (modelname.BGT)
- ASCII listing file (modelname.LST)

The binary files contain head, drawdown, and flow exchange values for each grid cell, while the ASCII listing file contains all relevant information on the operation of MODFLOW, and the simulation results. The listing (.LST) file is useful if errors occur during a simulation and you want to know how far MODFLOW progressed, or if you want to examine head or drawdown values at given intervals.

For a steady-state simulation, only one set of values for each grid cell are written to these files. However, for transient simulations, each grid cell may contain simulation results for each time step, resulting in file that can become unnecessarily large. By default, the information is saved in the binary files at the end of each stress period, and at the end of the simulation in the listing (.LST) file.

The first two columns list the available stress periods and associated time steps for the entire simulation (only one stress period and time step will be listed for steady-state simulations). The remaining columns indicate the information which can be written and saved to the various MODFLOW output files. To select an output option, click in the appropriate checkbox and a checkmark (✓) will appear to indicate that the selected information will be written for the selected time step.

The columns labeled Save to Binary will save the output information to the binary files as described below.

- **Heads**: Saves the heads in the binary heads file (.HDS).
- **DDown**: Saves the drawdown in the binary drawdown file (.DDN).
- **F.Term**: Saves the cell-by-cell flow terms in the binary budget (.BGT) file.

**Please Note**: The Zone Budget program uses the.BGT file for calculating the flow between zones. Therefore, to change the frequency at which the Zone Budget information is saved, select the desired F.Term intervals.

The columns labeled Print to .LST will save the output information to the listing file as described below.

- **Heads**: Saves the heads in the listing file.
- **DDown**: Saves the drawdown in the listing file.
- **F.Term**: Saves the flux terms (cell by cell flow terms) in the listing file.
- **Budget**: Saves the budget information in the listing file.

**Please Note**: MODFLOW only allows the flow terms (F.Term) to be stored once, in either the binary budget file (.BGT) or the listing file (.LST). Be aware that this setting
can be lost if MODFLOW is being run together with MODPATH, because MODPATH requires the flow terms to be written to the .BGT file, and not to the .LST file.

The checkbox labeled Save.FLO file will save the cell-by-cell flow terms required by MT3D/RT3D, when neither MT3D nor RT3D is being run at the same time as MODFLOW.

**Saving Output Every Nth Time Step**

For simulations with many stress periods and time steps, it can be very tedious to manually select the desired output time step intervals. The row of fields underneath the Output Control table are used to specify regular time step intervals for saving files during Each N-th step in each stress period. The first text box is where the N value is entered. To apply this value to the column, click the underlying checkbox.

If a subsequent engine is included in the run formation (e.g. ZoneBudget, MODPATH) with the MODFLOW simulation, Visual MODFLOW Flex will save the flow terms for all time steps by default.

10.10.1.12 Advanced Settings

Advanced translation settings are available for flow simulations to enable more advanced control of translation and package settings. General and package-specific advanced settings are described below. General advanced settings are available for all Flow and Transport engines supported by Visual MODFLOW Flex, while certain engines have additional package-specific advanced settings available:

- General
- MODFLOW-2005/NWT
- MODFLOW-SURFACT

Advanced settings for SEAWAT warrant more detailed discussion and are presented separately for the **VDF** and **VSC** packages in individual sections.

**Advanced Settings - General**

The following settings are available for all packages:
• **CUnit**: The index for the package in the CUNIT array of the BAS file *read-only*

• **Ext**: The file extension of the package *read-only*

• **LUnit**: The file unit number to which the package will be read/written during the engine run *read-only*

• **Run**: Yes/No switch to determine whether the package will be added to the .NAM/.IN file and run during the simulation. **Note**: certain packages are required as part of a simulation (e.g. BAS file) and cannot be selected to not run.

• **Translate**: Yes/No switch to determine whether the package will be translated. **Note**: packages that do not have sufficient data will not be translated (e.g. a river package file will not be translated if no rivers are listed in the model explorer in the active numerical model workflow).

**Please Note**: If a supported package is *not* represented in the model, by default it will not be translated and run, even if translate and run are both set to "Yes". For example if there are no drains represented in the model, then the DRN package will not be translated and run.

**Custom/Pre-existing Input Files**

If you develop or modify a package input file outside of Visual MODFLOW Flex, you can still use it as part of your modeling workflow within the Flex environment. This is particularly useful if you want to use one or more packages that are not yet supported in Visual MODFLOW Flow or if you have custom methods/utilities to generate input files. To use a custom input file:

1. Set Run = Yes

2. Set Translate = No

3. Add the input file to the model run folder with the correct name and extension: [model].[ext]. The model run folder can be found by selecting the folder icon button, the model name will correspond to the name of your conceptual model, and the extension
will be the Ext value for the package in question. For example, if you will to run the drain return package (DRT), and you are running a model where:

- the Visual MODFLOW Flex project file is called SiteModel.amd,
- the Conceptual Model is called Concept1,
- the Grid is called NumericalGrid,
- the Run is called Run 1, and
- the Flow Engine is MODFLOW-2005

then the DRT file must be saved as:

```
\SiteModel.data\MODFLOW\NumericalGrid\Run1\MODFLOW-2005\Concept1.DRT
```

If the above conditions are met, the custom package file will inserted into the relevant .NAM/.IN file and run with the model. Note that certain results from unsupported packages will not be read back into Visual MODFLOW Flex, such as the flow balance terms associated with the SFR package.

**Advanced Settings - MODFLOW-2005/NWT**

Package-specific settings for MODFLOW-2000, -2005, and -NWT are described below:

**CHD Settings**

The following settings are specific to the specified head (CHD) package:

- **Duplicate CHD Cell Filter** - When working with conceptual models, in some cases multiple constant head cells may be assigned to the same cell. When MODFLOW formulates the system of equations, it simply adds boundary conditions - for all other boundary conditions this is usually not a problem as these values still require a head solution for that cell; however adding the constant head values together typically does not produce the desired result. Therefore, Visual MODFLOW Flex includes an optional filter to handle these cases on translation:
  
  - Minimum - the minimum specified head value for the current stress period in the cell(s) with multiple values will be assigned
  - Maximum - the maximum specified head value for the current stress period in the cell(s) with multiple values will be assigned
  - Average [Default] - the arithmetic mean of specified head values for the current stress period in the cell(s) with multiple values will be assigned
  - Sum - specified head values values for the current stress period in the cell(s) with multiple values will be added.
OC Settings
The following settings are specific to the Output Control (OC) package:

- **Output control file format type** - An integer value used to define which cell in a vertical column that recharge and discharge is simulated.
  - use codes - the output control file will be generated with integer codes to that mark what output will be generated for each time step
  - use keywords [Default] - the output control file will be generated with keywords that are more easily legible. This option also enables the use of the compact budget format.

- **Use compact budget** - an optional keyword that controls the format of the output binary file
  - Yes [Default] - the compact budget format will be used. This format uses less disk space than the standard format developed with MODFLOW-88
  - No - the standard format developed with MODFLOW-88 will be used.

**Please Note:** because of the expected input formats for MODPATH and MOD-PATH3DU, the OC settings will be ignored when running particle tracking simulations - MODPATH requires the code format with standard output and MOD-PATH3DU requires the use of keywords and the compact budget format.

UZF Settings
The following settings are specific to the UZF package:

- **Unsaturated Zone Connection Option (NUZOPT)** - An integer value used to define which cell in a vertical column that recharge and discharge is simulated.
- Recharge to and discharge from only the top model layer. This option assumes land surface is defined as top of layer 1.
- Recharge to and discharge from the specified layer in variable IUZFBND. This option assumes land surface is defined as top of layer specified in IUZFBND.
- Recharge to and discharge from the highest active cell in each vertical column. Land surface is determined as top of layer specified in IUZFBND. A constant head node intercepts any recharge and prevents deeper percolation.

- **Vertical Hydraulic Conductivity Option** (IUZFOPT) - An integer value equal to 0, 1, or 2.
  - 0 - flow will not be routed through the unsaturated zone, rather infiltration will be applied directly to the saturated zone.
  - 1 - vertical hydraulic conductivity will be specified within the UZF1 Package input file using array VKS. [not supported]
  - 2 - vertical hydraulic conductivity will be specified within either the BCF or LPF Package input file

- **Water Routing** (IRUNFLG) - An integer value that specifies whether ground water that discharges to land surface will be routed to stream segments or lakes as specified in the IRUNBND array (IRUNFLG not equal to zero) or if ground-water discharge is removed from the model simulation and accounted for in the ground-water budget as a loss of water (IRUNFLG=0). The Streamflow-Routing (SFR2) and(or) the Lake (LAK3) Packages must be active if IRUNFLG is not zero.

- **Evapotranspiration** (IETFLG) - An integer value that specifies whether or not evapotranspiration (ET) will be simulated. ET will not be simulated if IETFLG is zero, otherwise it will be simulated.

- **Unsaturated Budget Output Method 1** (IUZFCB1) - An integer value used as a flag for writing ground-water recharge, ET, and ground-water discharge to land surface rates to a separate unformatted file using subroutine UBUDSV. If IUZFCB1>0, it is the unit number to which the cell-by-cell rates will be written when “SAVE BUDGET” or a non-zero value for ICBCFL is specified in Output Control. If IUZFCB1 ≤ 0, cell-by-cell rates will not be written to a file.

- **Unsaturated Budget Output Method 2** (IUZFCB2) - An integer value used as a flag for writing ground-water recharge, ET, and ground-water discharge to land surface rates to a separate unformatted file using module UBDVS3. If IUZFCB2>0, it is the unit number to which cell-by-cell rates will be written when “SAVE BUDGET” or a non-zero value for ICBCFL is specified in Output Control. If IUZFCB2≤0, cell-by-cell rates will not be written to file.

- **Number of Trailing Waves** (NTRAIL2) - An integer value equal to the number of trailing waves used to define the water-content profile following a decrease in the infiltration rate. The number of trailing waves varies depending on the problem, but a range between 10 and 20 is usually adequate. More trailing waves may decrease mass-balance error and will increase computational requirements and memory usage. This variable is not specified if IUZFOPT is zero or less.
- **Number of Wave Sets** (NSETS2) - An integer value equal to the number of wave sets used to simulate multiple infiltration periods. The number of wave sets should be set to 20 for most problems involving time varying infiltration. The total number of waves allowed within an unsaturated zone cell is equal to NTRAIL2 * NSETS2. An error will occur if the number of waves in a cell exceeds this value. This variable is not specified if IUZFOPT is zero or less.

- **Average Undulation Depth** (SURFDEP) - A real value equal to the average undulation depth within a finite-difference cell.

### Advanced Settings - MODFLOW-SURFACT

#### Basic Flow (BAS) Package Settings

The following settings are specific to the BAS package:

- **IDUAL** - An integer value used to define the dual-domain model for transport simulations:
  - If IDUAL = 0, the simulation is for a single domain continuum.
  - If IDUAL = 1, the simulation is for a dual domain representation with no equilibrium sorption occurring in the immobile domain.
  - If IDUAL = 2, the simulation is for a dual domain representation with equilibrium sorption characteristics of the immobile domain being equal to that of the mobile domain at any node in the system.
  - If IDUAL = 3, the simulation is for a dual domain representation with equilibrium adsorption characteristics of the immobile domain being distinct from those of the mobile domain, requiring additional input of these properties.

- **ITRAN** - An integer flag that determines if the simulation is either flow only or flow and transport:
  - If ITRAN = 0, the simulation is for flow only.
  - If ITRAN = 1, the simulation is for both flow and transport.

The above settings will override the Modeling Objectives selections and can be used to simplify the model without affecting the values stored in at the Modeling Objectives step or in the model explorer.

**Please Note**: you will likely encounter errors running the model if you select a higher value of IDUAL or ITRAN than you have set up your model to handle, for example: selecting ITRAN=1 without first having set up transport species.

#### Block-Centered Flow (BCF4) Package Settings

The following settings are specific to the BCF4 package:
- **ATMGP** - Standard atmospheric pressure
- **COMPAIR** - Compressibility of air
- **COMPWATER** - Compressibility of water
- **GRAV** - Gravitational acceleration constant
- **IREALSL** - is a flag indicating which soil moisture functions are used to define flow in the unsaturated zone above the water-table:
  - $\text{IREALSL} = 0$: Pseudo-soil relations are used
  - $\text{IREALSL} > 0$: Flow in the vadose is treated rigorously
  - $\text{IREALSL} = 1$: Groundwater flow is simulated in the vadose zone and the van Genuchten functions are used for retention and relative permeability characteristics of the unsaturated layer(s).
  - $\text{IREALSL} = 2$: Groundwater flow is simulated in the vadose zone and the van Genuchten function is used for retention and the Brooks-Corey function is used for relative permeability characteristics of the unsaturated layer(s).
  - $\text{IREALSL} = 3$: Soil vapor flow is simulated in the vadose zone and the van Genuchten functions are used for retention and relative permeability characteristics of the unsaturated layer(s).
  - $\text{IREALSL} = 4$: Soil vapor flow is simulated in the vadose zone and the van Genuchten function is used for retention and the Brooks-Corey function is used for relative permeability characteristics of the unsaturated layer(s).

- **RHOAP** - Density of air at reference pressure.
- **RHOWP** - Density of water.
- **VISG** - Viscosity of air.
- **VISW** - Viscosity of water

**Basic Transport (BTN) Package Settings**
The following settings are specific to the BTN package:

- **LINR** - is an integer flag indicating which retardation model will be used:
  - 0 - retardation is *not* applied
  - 1 - linear retardation is applied
  - 2 - non-linear retardation is applied using the Freundlich isotherm sorption model
Recharge-Seepage Face (RSF4) Package Settings
The following settings are specific to the RSF4 package:

- **IUNCNF** - is an integer flag for the unconfined option:
  - 0 - the specified recharge (RECH) is applied always like in a confined case even if the calculated water table exceeds the top elevation in layer 1.
  - 1 - Recharge seepage face boundary condition is employed and recharge (RECH) will be applied until water table reaches the ponding elevations described in PNDEL array.

- **NRCHOP** - is an integer flag for the recharge option. Recharge rates are defined in a two-dimensional array (RECH) with one value for each vertical column. Accordingly, recharge is applied to one cell in each vertical column, and the option code determines which cell in the column is selected for recharge. NRCHOP can have the following values:
  - 1 - Recharge is only applied to the top grid layer
  - 2 - Vertical distribution of recharge is specified in array IRCH [option not currently supported]
  - 3 - Recharge is applied to the highest active cell in each vertical column. Note: a constant-head node intercepts recharge and prevents deeper infiltration

Time-Varying Material Properties (TMP1) Package Settings
The following settings are specific to the TMP1 package:

- **ICRREAD** - is a switch for reading/using the vertical hydraulic conductivity (Kzz) scaling information (Cannot be used with ICVREAD).
- **ICVREAD** - is a switch for reading/using the vertical leakance scaling information. (Cannot be used with ICRREAD).
- **IHYREAD** - is a switch for reading/using the horizontal hydraulic conductivity (Kxx, Kyy) scaling information
- **ISC1READ** - is a switch for reading/using the specific storage (Ss) scaling information
- **ISC2READ** - is a switch for reading/using the specific yield (Ss) scaling information

⚠️ Please Note: if ISC1READ/ISC2READ are scaled, IHYREAD is required (even if the scaling values are all set to 1).

10.10.2 MODPATH

The MODPATH program is used for calculating the advective flow pathlines for forward tracking and backward tracking particles. MODPATH has run-time settings for Discharge Options which are used to tell MODPATH what to do with particles when they enter a grid cell where water is leaving the system. Any grid cell where water is leaving the system is
classified as a “sink”. For example, an extraction well is a sink, or all cells in layer 1 with evapotranspiration are sinks.

**Discharge Options**

**Weak Sink Options**

In many cases the water leaving the system through a grid cell is less than the amount of water entering the grid cell. If this difference is small, the cell is classified as a “weak sink”. Since MODPATH cannot always determine if a particle should be removed from the system when it encounters a weak sink, there are three options to control how particles should be treated when sinks are encountered.

- Particles always pass through cells with weak sinks
- Particles are always stopped when they enter cells with internal sinks
- Particles are stopped in the cells where discharge to sinks is greater than a specified total inflow to the cell [default = 5%].

The desired Weak Sink Option may be selected from the Discharge Options window shown in the following figure.

![Discharge Options Window](image)

**Recharge/EVT Options**

The Recharge Options are used to define how MODPATH treats the Recharge flow entering the system. The Recharge Options are:

- Recharge flux is treated as internal sources and sinks for all cells
- Recharge flux is assigned to the top face of all cells

The first option treats recharge as a distributed source entering the cell from all sides, while the second option treats recharge as though it is entering only through the top face of the cell.

According to the MODPATH reference manual, the distributed source approximation for areal recharge is usually only appropriate for two-dimensional areal flow models.
The Evapotranspiration Options are the same as described for Recharge above.

Please Note: as of Visual MODFLOW Flex version 6.0, the reference time settings have been moved to the Define Particles step so that release times can be assigned distinct values for each particle group.

10.10.3 MT3DMS/RT3D

MT3DMS and RT3D share the same set of available translation settings in Visual MODFLOW Flex since RT3D is effectively a modified version of MT3DMS and shares a common code base. The main differences are that MT3DMS provides more advanced capabilities for simulating dual-domain/sorption modeling while RT3D provides more advanced capabilities for simulating more complex bio-chemical reactions.

The following sections describes the settings for adjusting the solver, advection method, output times, and miscellaneous settings.

- General Settings
- Solution Method
- Output Control
- Advanced Settings

Please Note: SEAWAT is a coupled variable-density flow and transport model that is based on linking modified versions of the MODFLOW-2000 and MT3DMS engines. As such, the translation settings related to transport for MT3DMS also apply to SEAWAT. For more information on the SEAWAT-specific packages see the SEAWAT section.

10.10.3.1 General Settings

Visual MODFLOW Flex supports MT3DMS v.5.1 and RT3D v2.5. The following sections describes the settings for adjusting the solver, advection method, output times, and miscellaneous settings. The translation settings for MT3DMS v5.1 and RT3D v2.5 have the same set of available translation settings in Visual MODFLOW Flex since these two transport engines share a RT3D is effectively a modified version of MT3DMS and shares a common code base. The main difference is that MT3DMS provides more advanced capabilities for simulating dual-domain/sorption modeling while RT3D provides more advanced capabilities for simulating more complex bio-chemical reactions.
Porosity Options
The Porosity options are used to select which porosity measurement to use for the transport solution. For advection-dominated transport, the best choice is to apply the "Effective" porosity option, then diffusion into and out of dead-end pore spaces can be considered negligible. For diffusion dominated transport, the best choice is to select the "Total" porosity option to account for mass transfer to and from dead-end pore spaces.

Courant Number
The Courant number represents the number of cells a particle will be allowed to move through in any direction, in one transport step, when the MOC, MMOC and HMOC methods are used. Generally, the Courant Number is between 0.5 and 1.0; however, values in excess of 1.0 can be used with caution. If the upstream finite-difference method is used, the Courant Number must not exceed 1.0. Since the flow terms in all cells in the entire grid are checked when determining the maximum allowable step size for particle tracking, there may be some cells outside the area of interest with high flow rates. The high flow rates in these cells will control the time step calculation in MT3D. In this situation, setting the Courant Number greater than 1 will not affect the accuracy of the simulation.

Minimum Saturated Thickness as a Fraction of Cell Thickness
The Min. sat. thickness parameter is used to set a value for the minimum thickness of the saturated layer for each cell. This option is particularly important when active cells are running dry. This variable will be in the units specified for length during the initial setup of the model.

10.10.3.2 Solution Method
Select the solution method used to solve the advection-dispersion transport equation.
For numerical solution of the advection-dispersion transport equation, Visual MODFLOW Flex provides the following Solution Methods for the MT3D/RT3D/SEAWAT transport engines:

- Upstream Finite Difference (UFD)
- Central Finite Difference (CFD)
- Higher-order finite-volume TVD method.

No single solution method has been shown to be effective for all transport conditions. The combination of these solution methods, each having its own strengths and limitations, is believed to offer the best approach for solving the most wide-ranging transport problems with desired efficiency and accuracy. A brief description of all the above solution methods, and their advantages and disadvantages, is available below. Further, Zheng and Bennett (1995) provides an introduction to all these solution methods, and a discussion and comparison of their relative strengths and limitations with emphasis on their implications in solving practical problems.

**General methodologies of the solution methods**

MT3DMS is a transport model for simulating advection, dispersion, and chemical reactions of contaminants in groundwater flow systems. It solves the transport equation after the flow solution has been obtained from groundwater flow model (MODFLOW). The general advective-dispersive equation describing the fate and transport of contaminant of species $k$ in three-dimensional transient groundwater flow systems is:
\[
\frac{\partial (\Theta C^k)}{\partial t} = \frac{\partial}{\partial x_i} \left[ \Theta D_{ij} \frac{\partial C^k}{\partial x_j} \right] - \frac{\partial}{\partial x_i} \left( \Theta v_i C^k \right) + q_s C_s^k + \sum R_n
\]

where:

- \( C_k \) is the dissolved concentration of species \( k \)
- \( \Theta \) is the porosity of the subsurface medium
- \( t \) is time
- \( x_i \) is the distance along the respective Cartesian co-ordinate axis
- \( D_{ij} \) is the hydrodynamic dispersion coefficient tensor
- \( v_i \) is the seepage or linear pore water velocity. It is related to the specific discharge or Darcy flux through the relationship, \( \nu = q_s / \Theta \)
- \( q_s \) is the volumetric flow rate per unit volume of aquifer representing fluid sources (positive) and sinks (negative)
- \( C_s^k \) is the concentration of the source or sink flux for species \( k \)
- \( \sum R_n \) is the chemical reaction term.

All the solution methods listed above treat the dispersion, sink/source, and reaction terms in exactly the same fashion, using the block-centered finite-difference method, either explicit or implicit in time-weighting. They differ, however, in the way the advection term is solved. For instance:

- When the particle-based methods (MOC, MMOC, and HMOC) or the TVD method are selected to simulate solute transport, the transport equation is split into two parts. On the left-hand side are the mass accumulation term and the advection term (in fluid mechanics literature, the sum of these two has been referred to as the total derivative of the concentration with respect to time, \( DC/DT \)). On the right-hand side, the dispersion, reaction, and sink/source terms are represented with finite-difference approximation.
- When the finite-difference methods (UFD, CFD) are used, all terms in the governing equation are treated simultaneously, with all advection, dispersion, reactions, and sink/source terms represented with finite-difference approximations. The finite-difference solution is explicit or implicit-in-time weighting, depending on whether the Generalized Conjugate Gradient Solver (GCG) package is selected or not.

**Solution Method**

**Upstream Finite Difference Method (UFD):** Since, the finite-difference method does not involve particle tracking or concentration interpolations, it is normally more computationally efficient than the Method of Characteristics (MOC). In addition, the finite difference method normally has very small mass balance errors because it is based on the principle of mass
conservation. However, the Upstream Finite-Difference method can lead to significant numerical dispersion for problems having sharp concentration fronts.

**Central Finite Difference Method (CFD):** The central finite difference method does not exhibit the numerical dispersion problems like the Upstream Finite Difference method, but is susceptible to excessive artificial oscillations in advection dominated problems.

**Third-order Total-Variation-Diminishing Method (TVD):** The TVD scheme, which is mass conservative, solves the advection term based on the Universal Limiter for Transient Interpolation Modeling of the Advective Transport Equations (ULTIMATE) algorithm by Leonard (1988). As in the particle-based methods, the TVD scheme solves the advection component independent of the other terms in the transport equation. Results from the TVD scheme may exhibit minor numerical dispersion and minor oscillations in problems having sharp concentration fronts. Since the algorithm is explicit, there is a stability constraint on the step size. The maximum allowed value for the time step is the minimum time step calculated for every active cell.

**Please Note:** Modifying the default parameter settings for each Solution method requires some understanding of the techniques used to solve the advection-dispersion component of the transport equation. For more details, refer to the MT3DMS User’s manual (Zheng and Wang, 1999).

### Comparison of Solution Methods

<table>
<thead>
<tr>
<th>Advection Method</th>
<th>Advantages</th>
<th>Disadvantages</th>
<th>Appropriate Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Implicit Finite Difference (with Implicit GCG solver)</td>
<td>- Computationally efficient -Minimal mass balance errors -Faster than the explicit method</td>
<td>Numerical dispersion</td>
<td>Dispersed fronts Peclet number &lt;2</td>
</tr>
<tr>
<td>Explicit Finite-Difference (without Implicit GCG solver)</td>
<td>- Computationally efficient -Minimal mass balance errors</td>
<td>Numerical dispersion</td>
<td>Dispersed fronts Peclet number &lt;2</td>
</tr>
</tbody>
</table>
Using the GCG Solver

When the GCG Solver Package is specified to be included in a simulation, MT3D will do the following:

- When the finite-difference methods (UFD, CFD) are used, all terms in the governing equation are represented with implicit-in-time weighted finite-difference approximations.

- When the TVD method (or the particle-based methods MOC, MMOC and HMOC) are used to simulate solute transport, the terms on the right-hand side are represented with implicit-in-time weighted finite-difference approximations. The TVD algorithm implemented in MT3D is explicit-in-time. This means that even if the GCG Package is used, the time steps in the TVD run will still be subject to a time constraint - but only the advection term.

If the GCG solver is not included, then all terms are represented with explicit-in-time weighted finite-difference approximations.

When the GCG Solver is included, during translation the GCG package file (projectname.GCG) will be created in the model run directory.

**GCG Solver Settings**

Maximum number of outer iterations (MXITER) [Default = 1]: The outer loop in the iteration process updates all the coefficients that are concentration-dependent. The default number of outer iterations is one. It should be set to an integer greater than one only when a non-linear sorption isotherm is included in simulation. For further detail please refer to page 4-19 in the MT3DMS Documentation and User’s Guide.

Maximum number of inner iterations (ITER1) [Default = 50]: The inner loop in the iteration process continues to iterate toward the solution until the convergence criterion is obtained or the maximum number of inner iterations is reached. During these iterations, the coefficient matrix and the right hand side vector matrix remain unchanged. The default number of inner iterations is fifty. A maximum value of thirty to fifty inner iterations should be sufficient for most problems. For further details please refer to page 4-19 in the MT3DMS Documentation and User’s Guide.

The Relative convergence criterion (CCLOSE) [Default = $10^{-4}$]: is used to judge the convergence, in terms of relative concentration, of the inner iterations of the solver. The default value is $10^{-4}$. A value between $10^{-4}$ and $10^{-6}$ is generally adequate.
The Concentration change printing interval (IPRGC) [Default = 0]: is the interval for printing the maximum concentration changes of each iteration. The default is 0 for printing at the end of each stress period.

**Preconditioners**

The GCG Solver implicitly solves the non-advective terms in the transport equation using a two-iteration loop process and the choice between three pre-conditioning options:

- Jacobi (ISOLVE=1)
- SSOR (ISOLVE=2)
- Modified Incomplete Cholesky (ISOLVE=3)

The Modified Incomplete Cholesky method usually converges faster than the other two method options, but it requires significantly more memory. If SSOR is chosen, you must specify a relaxation factor.

**Dispersion tensor cross terms**

The GCG Solver has two options for dealing the cross-derivative terms of the dispersion tensor:

- Lump all dispersion cross terms to the right hand side (NCRS=0) will place all the dispersion tensor cross terms to the right hand side of the matrix equation and consider them as known values. This method approximates the equation (the loss of accuracy is generally insignificant) but is highly efficient (reduces the memory requirement by nearly two thirds) since there are significantly less unknown terms to be solved. For further detail please refer to the MT3DMS Documentation and User’s Guide.

- Include full dispersion tensor (NCRS=1) will keep all the dispersion tensor cross terms on the left-hand side of the matrix equation and consider them as unknown terms. This method gives the exact matrix equation but is memory intensive since there are significantly more unknown terms to be solved. For details, refer to the MT3DMS Documentation and User’s Guide.

**Initial Step Size (DT0):**

[Default = 0]: The Initial Step Size is used to override the automatically calculated initial time step. The user can assign a value to be used for the first iteration of the solution. The default value of 0 means that the MT3D calculated value will be used.

**Please Note:** If the implicit solver is not selected, then this value will be ignored and the value for DT0 will be taken from the Output Control settings.
Maximum transport step size (TTSMAX)

[Default = 0]: This value is the maximum time step size allowed when a multiplier greater than 1 is used. Setting this equal to zero imposes no maximum limit.

Multiplier (TTSMULT)

[Default = 1]: This is the multiplier used to calculate the size of the next transport time step when the finite difference method is used with the implicit GCG solver. A value between 1 and 2 is generally adequate. Using a time step multiplier will degrade the solution at later times when the timestep becomes large, but can significantly decrease the amount of time needed for the solution. If one of the particle-based methods or the TVD method is used, this value will be ignored.

Overview of MT3D Solution Methods

MT3D offers two general options for simulating solute transport:

1. When the "pure" finite difference methods are used (FD), all terms in the governing equation are treated simultaneously, with advection, dispersion, reactions, and source-sink mixing terms represented with finite-difference approximations.
2. When the operator-splitting methods are used to simulate solute transport, the governing equation is split into two parts. On the left-hand side are the mass accumulation term and the advection term (in the fluid mechanics literature, the sum of these two is referred to as the total derivative of the concentration with respect to time, Dc/Dt). On the right-hand side, the dispersion, reactions, and source-sink mixing terms are represented with finite-difference approximations. The operator-splitting techniques implemented in MT3D are the particle-based methods (MOC, MMOC and HMOC) and a 3rd-order TVD method.

Treatment of the transport terms with respect to time

MT3D offers the user two options for evaluating the terms in the finite-difference approximations, explicit or implicit-in-time weighting:

1. Explicit-in-time weighting. With explicit-in-time, or forward-in-time weighting (FIT), the terms in the finite-difference approximations are evaluated at the previous time step. This was the only option available in versions of MT3D from v.1.1 to MT3D96. Explicit-in-time weighting has the advantage of not requiring the assembly, storage, and solution of a matrix. It requires the least amount of RAM of all feasible methods. In particular, it requires no additional RAM with respect to the corresponding MODFLOW model. This was an important requirement for the development of MT3D, since the code was developed on a PC, for implementation on PCs. We now live in a world in which RAM is inexpensive. However, when MT3D was developed in 1987-1988, additional RAM over 640 KB was very limited and very expensive. Explicit-in-time weighting has the disadvantage of not being
stable unless tight restrictions on the size of time steps are satisfied. In many cases, the size of the time steps can be relatively small - and may only get smaller as the spatial resolution of a model was increased. Furthermore, in versions of MT3D preceding MT3D96, the calculation of the overall maximum allowable step size was not correct, and a model could still experience severe problems although it seemed that all stability criteria were being satisfied. In theory, the maximum Courant number may be 1.0. However, because the stability requirements were not calculated correctly, in practice users often had to specify maximum Courant numbers significantly smaller than 1.0 to ensure stability.

2. Implicit-in-time weighting. With implicit-in-time, or backward-in-time weighting (BIT), the terms in the finite-difference approximations are evaluated at the current time step. This option was added with the introduction of MT3DMS and MT3D99. Implicit-in-time weighting has the advantage of being unconditional stable, and allows the use of relatively large time steps. The option requires the assembly, storage, and solution of a matrix, but these requirements are not as onerous as they were, both because additional RAM is relatively inexpensive, and because accurate and efficient matrix solvers are now widely available. The disadvantage of the method is that it is now possible to create transport models that require much more storage than the corresponding MODFLOW model. Although implicit-in-time weighting is unconditionally stable, its accuracy is not assured if the time steps are too large. In order to guarantee the accuracy, the maximum Courant number may be limited to 1.0. As a simulation proceeds and the concentration gradients diminish, this requirement may be relaxed.

Time-weighting options with the particular MT3D solution methods

1. When the "pure" finite difference methods are used (FD), all terms in the governing equation may be represented with either explicit or implicit-in-time weighted finite-difference approximations.

2. When the particle-based methods (MOC, MMOC and HMOC) and the TVD method are used to simulate solute transport, the terms on the right-hand side (the dispersion, reactions, and source-sink mixing) may be represented with either explicit or implicit-in-time weighted finite-difference approximations.

A potential source of confusion to MT3D users is the apparent absence of a switch for specifying explicit or implicit time weighting. It turns out that this switch is 'implied'. When the user specifies that the GCG Solver Package is to be included in a simulation, the user is telling MT3D to do the following:

1. When the "pure" finite difference methods are used (FD), all terms in the governing equation are represented with implicit-in-time weighted finite-difference approximations.

2. When the particle-based methods (MOC, MMOC and HMOC) and the TVD method are used to simulate solute transport, the terms on the right-hand side are represented with implicit-in-time weighted finite-difference approximations. At the risk of confusing the issue, I should add at this point that the TVD algorithm implemented in MT3D is explicit-in-time. This means that even if the GCG Package is used, the times steps in TVD run will be still be subject to a time constraint - but only the advection term.
If the user does not specify the use of the GCG package, then all terms are represented with explicit-in-time weighted finite-difference.

### 10.10.3.3 Output Control

The Output Settings and Output Times settings are used to define the length of the transport simulation and the times at which the results will be saved to the binary concentration file (.UCN).

**Simulation time** is the total length of the transport simulation in the specified time units. This must be greater than zero.

**Max # of transport steps** is the maximum number of transport steps for the simulation. Once the simulation has run through the specified Max # of transport steps the model will stop. This parameter is primarily used to limit the size of the output files generated.

**Specify max stepsize** sets the maximum step size to use for each transport time step. MT3D will use the smaller of: the internally calculated maximum timestep size based on the Courant number, or the maximum timestep size specified here. If the implicit GCG solver is used, this value is ignored and the value found in the GCG Solver options is used instead.

**Save concentration at observation point for every Nth time step** allows to filter the amount of observation data that is saved to the .OBS file; ideal for transient runs with large output files.

**Save one-line summary of mass budget for every Nth time step** allows to filter the amount of mass budget data that is saved to the .MAS file; ideal for transient runs with large output files.

The Save .CBM/.CCM file option must be checked if you want Mass Balance output to be saved (for example, to generate Zone Budget-Transport data).

### Output Times

In the grid shown below, you can enter the output times for which you want MT3DMS to save during the run. The simulation results are saved in the MT3D output *.OT file.
The toolbar above this grid provides the following options (from left to right)
- Add row to the grid
- Delete selected row(s)
- Copy selected rows to clipboard
- Paste from clipboard: ideal if you have times defined in a column in an Excel spreadsheet, or in a text file (one time entry per row)

10.10.4 SEAWAT

Visual MODFLOW Flex supports SEAWAT v.4.00.05 - which based on a tight coupling of MODFLOW-2000 and MT3DMS. As such, the translation settings are very similar to the settings for MODFLOW and MT3DMS and largely follow the same format:

- settings and packages related to flow can be found under the SEAWAT node (refer to MODFLOW settings for details).
- settings and packages related to solute transport can be found under the SEAWAT Transport node (refer to the MT3DMS/RT3D settings).
- the SEAWAT-specific variable density flow (VDF) and viscosity (VSC) package settings can also be found under the SEAWAT Transport node

ℹ️ Please Note: the VSC package is only compatible with LPF as the selected internal flow package.
SEAWAT Flow Engine Settings

- General
- Settings
- Solvers
- Recharge and Evapotranspiration
- Lakes
- Layer Types
- Cell Rewetting
- Initial Heads
- Anisotropy
- Output Control
- Advanced Settings

SEAWAT Transport Engine Settings

- General Settings
- Solution Method
- Output Control
- VDF Package
- VSC Package
10.10.4.1 VDF Package

The Variable Density Flow (VDF) package is an integral part of the SEAWAT engine as it defines the relationship(s) between fluid density and solute concentration(s). The VDF package is required to run a SEAWAT model simulation. Numerous options and settings are supported as part of the translation settings in Visual MODFLOW Flex and are described below.

Settings for the VDF package are divided into high-level settings that are always required as part of the VDF package and low-level settings that are required (or not) based on the high-level settings. High-level settings are located in the VDF group of the advanced package table, while low-level settings are displayed as needed in the form area below. For more complete explanation of the input settings for the VDF package, please refer to p16-19 of the SEAWAT V4 Manual (Langevin et al., 2007).

Variable Density - Theory

In SEAWAT, fluid density is estimated as a linear function of the concentration of one or more solute species, temperature, and pressure head:

\[
\rho = \rho_{(ref)} + \sum \frac{\delta \rho}{\delta C_i} \cdot \left[ C_i - C_{i\,(ref)} \right] + \frac{\delta \rho}{\delta T} \cdot \left[ T - T_{(ref)} \right] + \frac{\delta \rho}{\delta l} \cdot \left[ l - l_{(ref)} \right]
\]
where:

\[ \rho \] is the fluid density \([\text{M/L}^3]\)

\[ \rho_{(\text{ref})} \] is the reference fluid density \([\text{M/L}^3]\)

\[ C_i \] is the concentration of the \(i^{th}\) species \([\text{M/L}^3]\)

\[ C_{i(\text{ref})} \] is the reference concentration, or the concentration at which the fluid density = \(\rho_{(\text{ref})} [\text{M/L}^3]\)

\[ \delta \rho/\delta C_i \] is the density/concentration slope, or change in density per unit change in solute concentration of the \(i^{th}\) species \([-\text{]}\)

\[ T \] is the temperature \([\Theta]\)

\[ T_{(\text{ref})} \] is the reference temperature, or the temperature at which the fluid density = \(\rho_{(\text{ref})} [\Theta]\)

\[ \delta \rho/\delta T \] is the density/temperature slope \([\text{M/L}^3\cdot\Theta]\)

\[ \ell \] is the pressure head \([\text{L}]\)

\[ \ell_{(\text{ref})} \] is the reference pressure head, or the pressure head at which the fluid density = \(\rho_{(\text{ref})} [\text{L}]\)

\[ \delta \rho/\delta \ell \] is the density/pressure head slope \([\text{M/L}^4]\)

### Commonly used values for the VDF package

<table>
<thead>
<tr>
<th>Variable</th>
<th>Reference Value</th>
<th>Slope</th>
<th>Setting Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>1,000 kg/m³ *</td>
<td>n/a</td>
<td>VDF Translation Settings</td>
</tr>
<tr>
<td>Salt Concentration</td>
<td>0 mg/L</td>
<td>0.714</td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>25 °C</td>
<td>-0.375 kg/m³ / °C</td>
<td></td>
</tr>
<tr>
<td>Pressure Head</td>
<td>0 m</td>
<td>4.46x10⁻³ kg/m⁴</td>
<td>VDF Translation Settings</td>
</tr>
</tbody>
</table>

**Source:** Langevin et al. (2007)

*See Table A.2 in Viessman and Lewis (2003) or similar hydrology/engineering reference textbook for more precise values*
VDF High-Level Settings

**Density Option**
The density option is a flag (MT3DRHOFLG) that defines which species contribute to the relationship of concentration-derived variable fluid density. The options are:

- **Calculate density from salt only**: salt is assumed to be the first species as defined in the modeling objectives [MT3DRHOFLG=1] [default]
- **Calculate density from multiple species**: species defined with non-zero density slope values as defined in the modeling objectives will be included in the variable density calculations [MT3DRHOFLG=-1]
- **Uncoupled simulation using pre-calculated density**: density will be estimated using input values from a specified .UCN file [MT3DRHOFLG=0]

**Internodal Density Calculation Algorithm**
The Internodal Density Calculation Algorithm is a flag (MFNADVFD) that determines the method for calculating the internodal density values used to conserve fluid mass. The options are:

- central-in-space algorithm (MFNADVFD = 2)
- upstream-weighted algorithm (MFNADVFD = 0)

**Variable Density Water Table Correction**
This option represents a flag (IWTABLE) that is used to activate the variable-density water-table corrections (Guo and Langevin, 2002).

- correction applied (IWTABLE = 0)
- correction not applied (IWTABLE = 1)

**Initial Time Step**
A variable for the length of the first transport timestep, used to start the simulation if both of the following two conditions are met: 1) the IMT Process is active, and 2) transport timesteps are calculated as a function of the user-specified Courant number (the MT3DMS input variable, PERCEL, is greater than zero)

VDF Low-Level Settings
**Coupling Iteration Radio Selection (NSWTCPL)**

If the model simulation is setup to dynamically calculate variable density flow (i.e. MT3DRHOFGLG<>0), then this option sets the parameters that define the feedback link that between flow and transport caused by the relationship between solute concentrations and fluid density. The options are as follows:

- **Implicit coupling using DNSCRIT** - density is only (re)calculated:
  - at the first transport step of the simulation,
  - at the last transport step of a given MODFLOW timestep, or
  - when the maximum density change at a cell is greater than the fluid density convergence criteria

- **Explicit Coupling with one iteration** - flow and transport will be explicitly coupled using a one-timestep lag.

- **Explicit Coupling with multiple iterations** - flow and transport will be explicitly coupled using a one-timestep lag. Density will be recalculated up to is the maximum number of non-linear coupling iterations for the flow and transport solutions.

**Please Note**: The explicit coupling option is normally much faster than the iterative option and is recommended for most applications

**Maximum Number of implicitly coupling iterations (NWSTCPL)**

SEAWAT will stop execution if the maximum number of iterations has been reached and convergence between flow and transport has not occurred.

**Minimum/Maximum Fluid Density**

Calculated density values will be truncated if they fall below the minimum fluid density or above the maximum fluid density. If a value of zero is entered for the minimum and maximum fluid density, then SEAWAT will *not* truncate to a lower and an upper range of calculated density values, respectively.

**Reference Fluid Density**

The reference fluid density is the density at the reference concentration, temperature, and pressure. See Theory Section above.

**Fluid Density Convergence Criteria (DNSCRIT)**

The fluid density convergence criteria (DNSCRIT) is a user-specified density value [M/L$^3$]. If explicit coupling with multiple iterations is selected, then DNSCRIT is the convergence criterion, in units of fluid density [M/L$^3$], for convergence between flow and transport. If the maximum fluid density difference between two consecutive implicit coupling iterations is not
less than DNSCRIT, the program will continue to iterate on the flow and transport equations, or will terminate if NSWTCPL is reached. If implicit coupling is selected, DNSCRIT is the maximum density threshold, in units of fluid density. If the fluid density change (between the present transport timestep and the last flow solution) at one or more cells is greater than DNSCRIT, then SEAWAT will update the flow field (by solving the flow equation with the updated density field).

**Density vs Pressure Head Slope (DRHODPRHD)**

Density vs Pressure Head Slope (DRHODPRHD) is the slope of the linear equation of state that relates fluid density to the height of the pressure head (in terms of the reference density). A value of zero, which is typically used for most problems, inactivates the dependence of fluid density on pressure. See Theory Section above.

**Reference pressure head (PRHDREF)**

Reference pressure head (PRHDREF) is the reference pressure head. This value should normally be set to zero. See Theory Section above.

**Uncoupled Simulation Pre-calculated Density Options**

If the Density Option is set is set to uncoupled (MT3DRHOFLG=0), then you will have the option of using:

- the reference fluid density or
- density from a .UCN file

If choosing the latter option, you can enter or browse to .UCN file that contains the densities to be used. The .UCN file must be based on a grid of the same dimensions and have the same number of timesteps as the simulation in which it is to be used.

**10.10.4.2 VSC Package**

The Viscosity (VSC) package is the package in SEAWAT that defines the relationship between solute concentration(s) and temperature with viscosity. The VSC package is optional to run a SEAWAT model simulation and can be turned off by setting Run=No in the advanced settings, if desired. Numerous options and settings are supported as part of the translation settings in Visual MODFLOW Flex and are described below.
For a more complete explanation of the input settings for the VSC package, please refer to p20-22 of the [SEAWAT V4 Manual](Langevin, et. al., 2007). The theoretical basis of how viscosity is used in SEAWAT is discussed on page 6 of the SEAWAT V4 Manual.

**Please Note**: The VSC package can only be used with the LPF internal flow package.

---

**Viscosity - Theory**

Unlike fluid density, a linear approximation of viscosity does not adequately represent the effect of temperature over the range of typical/natural conditions on dynamic viscosity. For this reason, an alternative equation for dynamic viscosity has been implemented in SEAWAT:

\[
\mu = \mu_{(\text{ref})} + \sum ?\mu / ?C_i \cdot [C_i - C_i(\text{ref})] + \mu(T)
\]

where:

- \(\mu\) is the fluid dynamic viscosity\(^2\) [M/L·T]
\( \mu_{\text{(ref)}} \) is the reference fluid dynamic viscosity \([M/L \cdot T]\)

\( C_i \) is the concentration of the \( i^{th} \) species \([M/L^3]\)

\( C_{i \text{(ref)}} \) is the reference concentration, or the concentration at which the fluid viscosity 
\( = \mu_{\text{(ref)}} \) \([M/L^3]\)

\( \mu'/C_i \) is the viscosity/concentration slope, or change in viscosity per unit change in solute concentration of the \( i^{th} \) species \([L^2/T]\)

\( T \) is the temperature \([\Theta]\)

\( \mu(T) \) is the viscosity/temperature function - see below.

*Note:* The governing equations implemented in SEAWAT use dynamic viscosity. For simplicity, subsequent references to viscosity refer to dynamic viscosity (rather than kinematic viscosity).

Values for the viscosity/concentration slope for each species \( (\mu'/C_i) \) and the reference concentrations \( (C_{i \text{(ref)}}) \) are set at the Define Modeling Objectives Step (if variable density is the selected flow type). The available/published empirical relationships between viscosity and temperature are tabulated below:

<table>
<thead>
<tr>
<th>Option</th>
<th>Equation</th>
<th>( A_1 )</th>
<th>( A_2 )</th>
<th>( A_3 )</th>
<th>( A_4 )</th>
<th>( A_5 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear</td>
<td>( \mu(T) = \frac{\partial \mu}{\partial T} \cdot (T - T_{\text{(ref)}}) )</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Voss (1984)</td>
<td>( \mu(T) = A_1 \cdot A_2 \cdot \frac{A_3}{(T + A_4)} )</td>
<td>2.93</td>
<td>4 x 10^{-5}</td>
<td>10</td>
<td>248.3</td>
<td>133.15</td>
</tr>
<tr>
<td>Pawlowski (1991)</td>
<td>( \mu(T) = A_1 \cdot [A_2 + A_3 \cdot (T + A_4)]^{A_5} )</td>
<td>1.0 x 10^{-3}</td>
<td>1</td>
<td>1.551</td>
<td>2 x 10^{-2}</td>
<td>-20</td>
</tr>
<tr>
<td>Guo and Zhao (2005)*</td>
<td>( \mu(T) = A_1 \cdot T^{A_2} )</td>
<td>0.16</td>
<td>8</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
</tbody>
</table>

**Source:** after Langevin et al. (2007)

**NOTES:**

- all equations are based on base units of \( L=\text{meters}, M=\text{kilograms}, \Theta=\text{degrees Celsius} \)
Guo and Zhao (2005) is applicable to oil rather than water at a temperature range of \(5^\circ C \leq T \leq 170^\circ C\).

### Viscosity - Settings

#### Viscosity Option (MT3DMUFLG)

The viscosity option is a flag (MT3DMUFLG) that defines which species contribute to the relationship of concentration-derived variable fluid viscosity. The options are:

- **Calculate viscosity from salt only**: salt is assumed to be the first species as defined in the modeling objectives \([MT3DMUFLG=1]\) [default]
- **Calculate viscosity from multiple species**: species defined with non-zero viscosity slope values as defined in the modeling objectives will be included in the viscosity calculations. This option also enables more advanced options to model the effects of temperature on viscosity \([MT3DMUFLG=-1]\)
- **Use specified viscosity**: viscosity will be estimated using a default value or input values from a specified .UCN file \([MT3DMUFLG=0]\)

#### Minimum/Maximum Fluid Viscosity (VISCMIN/VISCMAX)

Calculated viscosity values will be truncated if they fall below the minimum fluid viscosity or above the maximum fluid viscosity. If a value of zero is entered for the minimum or maximum fluid viscosity, then SEAWAT will *not* truncate to a lower or upper range of calculated viscosity values, respectively.

#### Reference Viscosity (VSCREF)

The reference fluid viscosity is the viscosity at the reference concentration and temperature.

#### Viscosity vs. Temperature

If a species has been designated as the temperature species as part of a variable density simulation at the Define Modeling Objectives and the Viscosity Option is set to calculate viscosity from multiple species, then the viscosity vs. temperature options described in the theory section above will be shown:
In this case, you may select one of the four viscosity/temperature relationships: Simple linear, Voss (1984), Pawlowski (1991), or Guo and Zhang (2005). The table below will be populated with the associated number of fitting parameters ($A_1 - A_5$, as applicable) set to the respective defaults.

**Specified Viscosity Settings**

If the Viscosity Option is set is set to uncoupled (MT3DMUFLG=0), then you will have the option of using:

- the reference viscosity,
- read specified viscosity from a .UCN file
- read concentration from .UCN file and convert to viscosity using linear expression

If choosing either of the last two options, you can enter or browse to .UCN file that contains the file to be used for the respective option. The .UCN file must be based on a grid of the same dimensions and have the same number of timesteps as the simulation in which it is to be used.
10.11 Run Numerical Engines

One of the benefits of Visual MODFLOW is the integration of all of the numeric engines including:

- **Flow**: MODFLOW 2000, MODFLOW 2005, MODFLOW-NWT, MODFLOW-LGR, SEAWAT, or MODFLOW-SURFACT (MODFLOW-USG is handled through a separate workflow).
- **Transport**: MT3DMS, RT3D, SEAWAT or MODFLOW-SURFACT
- **Post-Processing**: MODPATH, and/or Zone Budget

**Please Note**: if SEAWAT is the selected flow engine, then it will also be a required transport engine as SEAWAT is a coupled flow and transport model. Similarly, MODFLOW-SURFACT is the selected flow engine and transport is active, then MODFLOW-SURFACT will be the only available transport engine as it is a coupled flow and transport model.

To run a simulation with any of the above numeric engines, be sure to select Run from the top menu bar of the Run section and the Engines to Run window will appear as shown in the following figure.

```
Available Flow Engines in the Finite Difference Numerical Model Workflow:

USGS MODFLOW 2000 from WH
USGS MODFLOW 2005 from WH
USGS MODFLOW-LGR from WH
MODFLOW-NWT
MODFLOW-SURFACT from HGL

Note: SEAWAT will only appear if the Flow type is set to variable density at the Define Modeling Objective step, in which case, it will be the only available flow/transport engine.
```
The MODFLOW engine can be run by itself without any requirements from any of the other engines.

The MODPATH engine requires a valid .BGT file to be present in the project folder. This file will be automatically created if MODFLOW and MODPATH are mutually selected.

The Zone Budget engine requires a valid .BGT file to be present in the project folder. This file will be automatically created if MODFLOW and Zone Budget are mutually selected.

The button will run the selected numeric engines using the required (translated) data files already present in the project folder. If the required files are not present in the project folder, the numeric engines will not run. A warning message will appear indicating that the Visual MODFLOW files will not be translated prior to running the model.

Press the button to start the simulation.
When the run completes, the Heads, Drawdown, Budget, and Water Table nodes will be added to the Output folder on the model tree. A saturation node will also be added if an unsaturated flow model is run (using MODFLOW-SURFACT or the UZF package). And one node will be added per simulated species for simulations including groundwater transport (e.g. using MT3D-MS, RT3D, SEAWAT, or MODFLOW-SURFACT with transport).

10.12 View Results

At the View Results step, you have the option to View Maps or View Charts.

At the View Maps step you can visualize output:

- heads, drawdown, budget, velocity;
- concentrations (if transport was simulated); and
- pathlines (if particle tracking was simulated).

View Charts contains several chart types:

- Observed vs. Calculated Heads
- Time Series
- Zone Budget
- Mass Balance.

10.12.1 Contours and Color Floods

Following a successful model simulation, outputs are generated and Outputs branch in the Model Explorer tree for the active run.

- Flow
  - Heads and Drawdown
At the View Maps step, you can view contours and color shading of the flow and transport results.

By default, you will see color shading and contours of the calculated heads, in layer view. You can display heads along a row, and along a column, and in 3D, using the same tools as you used earlier.

If your model is transient, use the time controls above the Flex Viewer to change the output time; as you do this, all active viewers (layer, row, column, 3D), will refresh to show the heads for the new output time.
The display settings (color classes, contour line intervals, etc.) for the Layer, Cross-section, or 3D views can be adjusted in the Style settings. To load these settings, right click on Heads or Drawdown in the Outputs section of the Model Explorer, and select “Settings”. For more details about these settings, see the section on 3D Gridded Data Settings.

**Budget**

Cell-by-cell flows read from the MODFLOW .BGT file can be viewed in a stand-alone 3D Viewer. Once displayed in the viewer, load the Style settings for Budget, and choose the desired display type (Cells is turned on by default). Under the Colors panel (shown below), you can select from various Budget attributes:

- **XRate**: lateral flow into the cell from the right and left face (adjacent columns)
- **YRate**: lateral flow into the cell from the front and rear face (adjacent rows)
- ZRate: vertical flow into the cell from cells in the upper and lower layer face (adjacent layers)
- InFlowRate: total flow into the cell from all adjacent cells
- OutFlowRate: total flow out of the cell from all adjacent cells
- NetFlowRate: InFlowRate - OutFlowRate

One term will be also added for each active boundary condition type with external flows in the model simulation (e.g. Wells, Drains, Recharge, Constant Heads, Rivers, etc.).

**Calculation Cell Flow–in/ Flow–out Based on Inter–Cell Flow Terms**

Inter–cell flow terms are saved in the output budget file (.BGT) for three neighbors of a given cell(j,i,k):

- RIGHT FACE \((j+\frac{1}{2}, i, k)\)
- FRONT FACE \((j, i+\frac{1}{2}, k)\)
- LOWER FACE \((j, i, k+\frac{1}{2})\)

*Indices for the six adjacent cell faces surrounding cell \(j,i,k\).*

The inter-cell flows for the remaining three faces are given by:

- LEFT FACE \((j-\frac{1}{2}, i, k) = RIGHT FACE([j-1]+\frac{1}{2}, i, k)\)
- FRONT FACE \((j, i+\frac{1}{2}, k) = REAR FACE(j, [i-1]+\frac{1}{2}, k)\)
- UPPER FACE \((j, i, k+\frac{1}{2}) = LOWER FACE(j, i, [k-1]+\frac{1}{2})\)*
Indices for the six adjacent cells surrounding cell \( j,i,k \).
Adapted from MODFLOW-2005 Users Manual, USGS Publication TM6-A16

InFlowRate (Flow In): total amount of flux into that cell. Calculated as follows:

\[
\text{InFlowRate} = \]
\[
\text{FLOW RIGHT FACE}[i, j-1, k] \times H(\text{FLOW RIGHT FACE}[i, j-1, k]) + \text{FLOW FRONT FACE}[i-1, j, k] \times H(\text{FLOW FRONT FACE}[i-1, j, k]) + \text{FLOW LOWER FACE}[i, j, k-1] \times H(\text{FLOW LOWER FACE}[i, j, k-1]) - \text{FLOW RIGHT FACE}[i, j, k] \times H(-\text{FLOW RIGHT FACE}[i, j, k]) - \text{FLOW FRONT FACE}[i, j, k] \times H(-\text{FLOW FRONT FACE}[i, j, k]) - \text{FLOW LOWER FACE}[i, j, k] \times H(-\text{FLOW LOWER FACE}[i, j, k])
\]

OutFlowRate (Flow Out): total amount of flux out of that cell. Calculated as follows:

\[
\text{OutFlowRate} = \]
\[
\text{FLOW RIGHT FACE}[i, j, k] \times H(\text{FLOW RIGHT FACE}[i, j, k]) + \text{FLOW FRONT FACE}[i, j, k] \times H(\text{FLOW FRONT FACE}[i, j, k]) + \text{FLOW LOWER FACE}[i, j, k] \times H(\text{FLOW LOWER FACE}[i, j, k]) - \text{FLOW RIGHT FACE}[i, j-1, k] \times H(-\text{FLOW RIGHT FACE}[i, j-1, k]) - \text{FLOW FRONT FACE}[i-1, j, k] \times H(-\text{FLOW FRONT FACE}[i-1, j, k]) - \text{FLOW LOWER FACE}[i, j, k-1] \times H(-\text{FLOW LOWER FACE}[i, j, k-1])
\]
Where \( H(x) \) is a known Heaviside Step Function; for more details on this function, see: [http://mathworld.wolfram.com/HeavisideStepFunction.html](http://mathworld.wolfram.com/HeavisideStepFunction.html)

\[
\text{NetFlowRate} = \text{InFlowRate} - \text{OutFlowRate}
\]

**Velocity**

At the View Maps step, you can view groundwater velocities in a variety of ways as described in the section on [3D Gridded Data Settings](#). By default, velocity vectors are shown as **vectors**.

---

**Calculation of Velocities in Visual MODFLOW Flex for Finite Difference Grids**

Model velocities are estimated in Visual MODFLOW Flex using a two-step process:

1. Estimate inter-cell velocities at each cell face in each component direction (X, Y, and Z)
2. Interpolate component velocities to the cell centroid
Inter-cell Darcy Velocity and Average Linear Velocity

Inter-cell Darcy velocities (also known as Fluxes) are calculated in Visual MODFLOW Flex using the following equations:

\[
DV_{X1} = \frac{Q_{(j-\frac{1}{2},i,k)}}{\Delta Y(i) \cdot \Delta Z(k)}
\]

\[
DV_{X2} = \frac{Q_{(j+\frac{1}{2},i,k)}}{\Delta Y(i) \cdot \Delta Z(k)}
\]

\[
DV_{Y1} = \frac{Q_{(j,i+\frac{1}{2},k)}}{\Delta X(j) \cdot \Delta Z(k)}
\]

\[
DV_{Y2} = \frac{Q_{(j,i-\frac{1}{2},k)}}{\Delta X(j) \cdot \Delta Z(k)}
\]

\[
DV_{Z1} = \frac{Q_{(j,i,k-\frac{1}{2})}}{\Delta X(j) \cdot \Delta Y(i)}
\]

\[
DV_{Z2} = \frac{Q_{(j,i,k+\frac{1}{2})}}{\Delta X(j) \cdot \Delta Y(i)}
\]

Inter-cell average linear velocities are estimated from the Darcy velocities:

\[
V_{ave} = \frac{DV}{\varphi_{(j,i,k)}}
\]

where \( \varphi_{(j,i,k)} \) is the effective porosity of the current cell

Component velocities interpolated to the cell centroid

Component velocities are interpolated to the cell centroid by the following equations:
Water Table

This is the elevation where the calculated head value is equal to the Water table elevation (pressure head equals zero). In 2D (Layer) and 3D views, the Water table is displayed as a surface object, and as such has style settings similar to Surfaces (for more details on these options, see Style Settings: Surfaces). In 2D cross-sectional view (by row or column), the water table is drawn as a solid line. On the toolbar above the view, you have an option to control how the line is calculated and displayed.

Smooth WTL (Water Table Line): the line is smoothed (interpolated) between each grid cell. Since the line is interpolated, this is an approximation.
Discrete WTL (Water Table Line): the line is drawn as piecewise constant, with a flat line drawn across each grid cell at the elevation corresponding to the water table. This option better reflects the actual output data and the MODFLOW grid cell geometry.

If your model is Transient, you can adjust the output time as explained above, and the Water table display will update to reflect the new values for that output time.

⚠️ The Water Table is currently only calculated for MODFLOW-2000, -2005, -NWT, -SURFACT, and SEAWAT model runs; it is not calculated for MODFLOW-LGR.

Export

The water table data can also be exported to .CSV file for further post-processing. Right-click on Water Table from the Model Explorer and select "Export..."

The format of this file is as follows:
- x (x-coordinate at center of cell)
- y (y-coordinate at center of cell)
- ztop (elevation of the top of layer 1; this can be used to calculate depth to water table)
- row
- column
- layer
- wt_"time": water table value at the defined MODFLOW time step; for transient models, you will see additional columns for each saved MODFLOW time step.

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>H</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>x</td>
<td>y</td>
<td>ztop</td>
<td>row</td>
<td>column</td>
<td>layer</td>
<td>wt_299.9999</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>-975</td>
<td>-1950</td>
<td>624.95</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>624.95</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>-925</td>
<td>-1950</td>
<td>624.95</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>624.95</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>-875</td>
<td>-1950</td>
<td>625.53</td>
<td>1</td>
<td>3</td>
<td>1</td>
<td>625.53</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>-825</td>
<td>-1950</td>
<td>625.53</td>
<td>1</td>
<td>4</td>
<td>1</td>
<td>625.53</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>-775</td>
<td>-1950</td>
<td>626.14</td>
<td>1</td>
<td>5</td>
<td>1</td>
<td>626.14</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>-725</td>
<td>-1950</td>
<td>626.14</td>
<td>1</td>
<td>6</td>
<td>1</td>
<td>626.14</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>-675</td>
<td>-1950</td>
<td>626.53</td>
<td>1</td>
<td>7</td>
<td>1</td>
<td>626.53</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>-625</td>
<td>-1950</td>
<td>626.86</td>
<td>1</td>
<td>8</td>
<td>1</td>
<td>626.86</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>-575</td>
<td>-1950</td>
<td>627.37</td>
<td>1</td>
<td>9</td>
<td>1</td>
<td>627.37</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>-525</td>
<td>-1950</td>
<td>627.72</td>
<td>1</td>
<td>10</td>
<td>1</td>
<td>627.72</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>-475</td>
<td>-1950</td>
<td>628.08</td>
<td>1</td>
<td>11</td>
<td>1</td>
<td>628.08</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>-425</td>
<td>-1950</td>
<td>628.45</td>
<td>1</td>
<td>12</td>
<td>1</td>
<td>628.45</td>
<td></td>
</tr>
</tbody>
</table>

**Concentrations**

If you ran a transport simulation (e.g. MT3D-MS, RT3D, or SEAWAT), you will also see output nodes for Concentration (one for each species). Only one 3D Gridded results can be shown in a 2D viewer at a given time, so in order to view Concentrations, you need to first turn off Heads or Drawdown from the model tree. (Note: the 3D viewer supports displaying multiple 3D Gridded results simultaneously; just create a new 3D viewer (Window / 3D Window from the main menu bar).
The settings allow you to clip the upper and/or lower levels - for example if we use the Lower Cutoff of 20:
This would result in the following rendering of the concentrations:
The option to use lower and upper cutoffs are available both for Colormap and ThreeSlices display options.

**Adjusting Style Settings**
The style settings of either the 2D or 3D views can be adjusted (note, each one must be defined independently)

First make the desired property group selected and visible in the Flex Viewer.

Next make the desired 2D viewer active

Then right-click on the desired property group in the tree as shown below and select `Settings`
The 2D Colors and Contour line settings can be adjusted. For more details, see Color and Contour settings.

**Exporting Outputs.**

Outputs can also be exported in a variety of formats. Please see the section on Exporting Files for more details.

**10.12.1.1 Pathlines and Time Markers**

Visual MODFLOW Flex includes options to adjust the Pathline travel time and time marker for a MODPATH or a MOD-PATH3DU particle track run (using either forward or backward particles).

If you ran a particle tracking simulation, you will see Pathlines appear as a new node (per particle group) in the tree under Output; add a check box beside the Pathlines to display these in the active 2D/3D Viewers.
The pathlines delineate the flow path and travel time of groundwater particles due to advective groundwater flow (as calculated by MODPATH/MOD-PATH3DU). The Pathlines Map is a little different from the other map types because it does not simply represent a "snap-shot" in time. Pathlines represent a historical travel log of the groundwater particles. Forward tracking pathlines are used to predict where groundwater is flowing, and how long it will take to reach a given location if it starts from a known location at a known time. Backward tracking pathlines are used to predict where groundwater at a given location and time is coming from, and how long it took to get there.

The "Pathlines Options" can be launched by right-clicking on Pathlines from the model explorer, and selecting "Pathline Options". The following window will appear (note that forward and backward pathlines can each have their own settings defined in the Options window.)
The Tracking time frame is used to select either Steady state pathlines (for all times), or to specify a maximum Travel time for all pathlines.

The Time markers frame is used to hide/show the pathline time markers (using the Visible checkbox), and to select the time intervals for each time marker. The Time markers can be plotted at Regular time intervals, or at Custom intervals selected by the user.

The pathline color and thickness can be adjusted in the Settings window; right click on the Pathlines object from the model explorer and select Settings.
Pathlines not showing up?

If forward tracking pathlines do not appear for some particles, check to make sure the particles are not assigned above the water table elevation, or in dry cells.

If backward tracking pathlines appear to ‘skim’ along the surface of the water table instead of exiting through recharge, check the MODPATH Discharge Options or the MOD-PATH3DU IFACE inputs in the translate section to make sure the recharge flux is being assigned to the top face of all cells instead of being treated as an internal source/sink.

Export

Pathline and time markers can be exported to shapefiles for further analysis; right-click on the Forward or Backward Pathlines on the model explorer, and choose Export. A points shapefile will be created for the time markers, and a polyline shapefile will be created for the pathlines. The exported files will utilize the pathline options you have currently defined in the view for the tracking time and time marker frequency.

10.12.2 Charts

In Visual MODFLOW Flex, upon completing a successful model run, you can view several different types of charts to review your results and to facilitate model calibration:

View Charts Workflow Step
- Calibration (calculated vs. observed),
- Time Series

Stand-Alone Charts
- Mass Balance
- Zone Budget

See the controls section below on how to create and modify the Charts associated with the View Charts workflow step, and the individual sections listed above for further details on the stand-alone charts.
Calibration (Calculated vs. Observed) Charts

The Calculated vs. Observed Chart consists of a scatter plot representing the comparison between the calculated and observed values at select points in time (as shown below). Simulated Heads or Concentrations can be compared to observed values and displayed on the Calculated vs. Observed Chart. For transient simulations containing many different observation times for each observation point, the quality of the model calibration will likely change throughout the simulation. Therefore, it is important to be able to evaluate the calibration at different times throughout the simulation.

Workflow Step Flow Model (Heads) Transport Model (Concentrations)

Unfortunately, the output times generated by MODFLOW rarely coincide with the actual times when the observed data was collected or recorded in the field. However, it is generally considered a good practice to interpolate the model calculated data to the observation times in order to compare the calculated vs. observed values. Visual MODFLOW Flex calculates the interpolated values for you; however, it is important to compare the model output times to the observations to assess the suitability of the interpolation relative to the time gap between output times and observations as well as relative to relevant stress periods and resulting changes in heads or concentrations.

The Scatter Graph of Calculated vs. Observed Values is the default Calibration Graph. This graph represents a snapshot in time of the comparison between the values calculated by the model (Y-axis), and the values observed or measured in the field (X-axis). The situation where all data points intersect the 45 degree line on the graph where X=Y represents an ideal calibration scenario, but it is not likely to happen when developing models of real-world sites.

If the data points appear above the X=Y line, then the calculated values are larger than the observed values, and the Calibration Residual is positive, indicating that the model is overpredicting. If the data points are under the X=Y line, then the calculated values are less than
the observed values, and the Calibration Residuals are negative, indicating that the model is under-predicting. The chart above indicates the model is over-predicting head values.

**Time Series Plots**

A Time-Series graph plots the value of one or more selected variable over time. In groundwater flow and contaminant transport modeling, time-series graphs are used to evaluate and compare temporal trends in the calculated Head, Drawdown, or Concentrations at selected Observation Points.

**Chart Controls**

Controls to adjust the charts are shown to the left of the chart.
- **Mass Balance**: Select this button to open the [Mass Balance Charts](#).

- **Zone Budget**: Select this button to open the [ZoneBudget Charts](#).

- **Parameter**: Select either Flow or Transport to display results from your flow or transport model, respectively.

- **Chart Type**: Select from either [Calc. vs. Obs.](#) or [Time Series](#).

- **Time**: Select listed times from the dropdown to display results at the specified output time step (*calculated vs. observed only*).

- **Labels**: Select the check box to show/hide labels for the plotted data points.

- **Observations Controls**
  - **All Times**: Select the check box to show all times or to only use the time specified in the **Time** dropdown list
  - **All Obs**: Select the check box to show all observations
- **Species Selection Box:** Only observations for the selected species will be shown (only shown when the Parameter option is Transport).
- **Observation Group Box:** Only observations for the selected observation group will be shown.
- **Layer Selection Box:** Only observations for the selected layers will be shown.
- **Observation Location Box:** Only observations for the selected locations will be shown.

- **Apply:** updates the chart using the selections in the Observations Controls.

**Statistics**

The Calibration Statistics are reported in the footer of the Calibration Plots window when the Calculated vs. Observed Scatter Graph is displayed. These statistics are calculated using the assumption that the calibration residuals are normally distributed. The individual statistics are described below:

### Calibration Residual

The Calibration Residual ($R_i$) is defined as the difference between the calculated results ($X_{cal}$) and the observed results ($X_{obs}$) at selected data points ($i \rightarrow n$) (as shown in the following equation):

$$R_i = X_{cal} - X_{obs}$$

The absolute value maximum and minimum residuals at the selected observation points are also reported.

### Residual Mean

The Residual Mean ($\bar{R}$) is a measure of the average Residual value defined by the equation:

$$\bar{R} = \frac{1}{n} \sum_{i=1}^{n} R_i,$$

where:

$n = \text{the number of observations}$

**Please Note:** there may be cases where over-calculated and under-calculated values will negate each other, and produce a Residual Mean value close to zero. This can lead to false
interpretation of the model calibration. The Residual Mean should never be used by itself as a measure of the fit between the simulated results and the observed data.

**Absolute Residual Mean**

The Absolute Residual Mean $|\bar{R}|$ is similar to the Residual Mean except that it is a measure of the average absolute Residual value defined by the equation:

$$|\bar{R}| = \frac{1}{n} \sum_{i=1}^{n} |R_i|$$

The Absolute Residual Mean measures the average *magnitude* of the Residuals, and therefore provides a better indication of calibration than the Residual Mean.

**Standard Error of the Estimate**

The Standard Error of the Estimate (SEE) is a measure of the variability of the residual around the expected residual value, and is expressed by the following equation:

$$SEE = \sqrt{\frac{1}{n-1} \sum_{i=1}^{n} (R_i - \bar{R})^2}$$

**Root Mean Squared**

The Root Mean Squared error (RMS) is defined by the following equation:

$$RMS = \sqrt{\frac{1}{n} \sum_{i=1}^{n} R_i^2}$$

**Normalized Root Mean Squared**

The Normalized Root Mean Squared is the RMS divided by the maximum difference in the observed head values, and is expressed by the following equation:

$$Normalized RMS = \frac{RMS}{(X_{obs})_{max} - (X_{obs})_{min}}$$
The Normalized RMS is expressed as a percentage, and is a more representative measure of the fit than the standard RMS, as it accounts for the scale of the potential range of data values.

For example, an RMS value of 1.5 will indicate a poor calibration for a model with a range of observed values between 10 and 20, but it will indicate an excellent calibration for a model with a range of observed values between 100 and 200. However, the Normalized RMS value for the first model would be 15%, while the Normalized RMS for the second model would be 1.5%. In the second situation, the Normalized RMS values clearly indicates the second model provides a good fit between the calculated and observed values.

**Correlation Coefficient**

The Correlation Coefficient \((Cor)\) is calculated as the covariance \((Cov)\) between the calculated results \((X_{cal})\) and the observed results \((X_{obs})\) at selected data points divided by the product of their standard deviations. The correlation coefficient is calculated using the following equation:

\[
Cor(X_{cal}, X_{obs}) = \frac{Cov(X_{cal}, X_{obs})}{\sigma_{cal} \cdot \sigma_{obs}}
\]

The covariance is calculated using the following equation:

\[
Cov(X_{cal}, X_{obs}) = \frac{1}{n} \sum_{i=1}^{n} (X_i - \mu_{cal})(X_i - \mu_{obs})
\]

Where \(\mu_{cal}\) and \(\mu_{obs}\) are the mean values of calculated and observed results, respectively.

\[
\mu_{cal} = \frac{1}{n} \sum_{i=1}^{n} X_{cal}
\]

\[
\mu_{obs} = \frac{1}{n} \sum_{i=1}^{n} X_{obs}
\]

The standard deviations are calculated by the equations:
Correlation Coefficients range in value from -1.0 to 1.0. The Correlation Coefficient determines whether two ranges of data move together - i.e. whether large values of one data set are associated with large values of the other data set (positive correlation), whether small values of one data set are associated with large values of the other data set (negative correlation), or whether values in both sets are unrelated (correlation near zero).

Export

Chart data can be exported to .CSV file, for further analysis (with Excel or other charting programs).

Just select the Export button from the toolbar when you are viewing either Calc. vs. Observed, or Time Series charts, for either Flow or Transport runs. All calibration data will be exported to the .CSV file (not just the selections you have defined in the Chart toolbox).

10.12.2.1 Mass Balance Charts

The finite difference formulation of the groundwater flow equation is based on conservation of mass, such that the mass of water entering the system through various sources should equal the mass of water leaving the system through the various sinks. The Mass Balance Graphs provide graphical representations of the mass of water (represented as volume, for MODFLOW models on the assumption of the constant density of water in the simulation domains) entering and leaving the system through the various sources and sinks.

The Flow Mass Balance Graphs plot the volume and rates of water entering and leaving the system through the flow boundary conditions, and from aquifer storage at the end of each stress period.

To plot the Flow Mass Balance Graphs, select the View Charts workflow step, click the Mass Balance Button on the upper left corner of the Chart view, and a Mass Balance: MODFLOW window will appear as shown below:
There are four different Flow Mass Balance Graphs:

- **Percent Discrepancy** plots the temporal changes in the Flow Mass Balance (total flow IN minus total flow OUT) expressed as a percentage of the total flow.

- **IN - OUT** plots the temporal change in the Flow Mass Balance (total flow IN minus the total flow OUT).

- **Time Series** plots the temporal flows IN and OUT of the system through the individual sources and sinks (flow boundary conditions and storage).

- **Time Step** plots a bar chart of the flow IN to the system and the flow OUT of the system through the individual sources and sinks (flow boundary conditions and storage) for selected Output Times.

**Note**: With both the Time Step and the Time Series graphs, it is important to remember that these graphs present the cumulative volume at a point in time, and not the volumetric loading rate at a point in time.

As mentioned above, the Flow Mass Balance Graphs can be used to plot the flow volume or flow rates of water coming IN and going OUT of the system. By default, the Mass Balance: MODFLOW window plots the flow volume mass balance.

Use the View option in the top menu bar to select either Rates or Volume options, or click the Cumulative volume (💧) or Rate (📊) icon in the toolbar.
**Sources and Sinks**

On the left-hand side of the Mass Balance: MODFLOW window is a list of all potential sources and sinks for the current flow model (boundary conditions not included in the model will not be listed). These sources and sinks are included in both the IN list and the OUT list. The IN list contains a summary of the flow coming into the system from each individual source term, while the OUT list contains a summary of the flow going out of the system through each individual sink term.

These controls can be used to add or remove the individual source and sink terms from either the Time Series graph or the Time step graph.

The Percent Discrepancy graph and the IN-OUT graph consider only the total flow into and out-of the entire system as a whole. As a result, the settings for the individual IN and OUT terms do not influence the results and appearance of these graphs.

**Arranging the Graphs**

Each of the Flow Mass Balance Graphs can be maximized to occupy the entire Mass Balance window by clicking the maximize button in the top right-hand corner of each graph window.

Alternately, the graphs can be tiled in the Mass Balance window by selecting Window/Tiles from the top menu bar.

### 10.12.2.2 ZoneBudget Charts

ZoneBudget is a program developed by the U.S. Geological Survey to calculate water budgets for user-defined zones in the model. The user-defined ZoneBudget zones are created during the Define Zone Budget Zones workflow step. The Zone Budget program calculates the flow mass balance for each user-defined zone.

**Flow Zone Budget Graphs**

The Flow Zone Budget Graphs plot the flow rates of water entering and leaving the user-defined zones through the flow boundary conditions, from aquifer storage, and through other user-defined zones.

To plot the Zone Budget Graphs, select the View Charts workflow step, click the Zone Budget Button on the upper left corner of the Chart view, and a Zone Budget: MODFLOW window will appear as shown below:
There are four different Flow Zone Budget Graphs:

- **Percent Discrepancy** plots the temporal changes in the Flow Mass Balance for the selected zone (total flow IN minus total flow OUT) expressed as a percentage of the total flow in the zone.
- **IN - OUT** plots the temporal change in the Flow Mass Balance for the selected zone (total flow IN minus the total flow OUT).
- **Time Series** plots the temporal flows IN and OUT of the selected zone through the individual sources and sinks (flow boundary conditions and storage).
- **Time Steps** plots a bar chart of the flow IN to the system and the flow OUT of the selected zone through the individual sources and sinks (flow boundary conditions and storage) for selected Output Times.

**Please Note**: With both the Time Steps and the Time Series graphs, it is important to remember that these graphs present the rate(s) at a point (or points) in time, and not the cumulative volume at a point in time.

**Controls**

On the left-hand side of the Zone Budget: MODFLOW window is a list of the Zone Budget zones available for the current model. The Zone Budget graphs described above will display the Zone Budget data for the selected Zone Budget zone.
The Zone List Box Control

The Zone Control indicates the current Zone for the active charts and the list box below shows all available ZoneBudget zones in the model. Clicking directly on a Zone in the list will switch to that Zone.

The IN/OUT List Box Controls

The IN list contains each of the possible source terms which may contribute flow to the selected Zone Budget zone, while the OUT list contains each of the possible sink terms which may divert flow out of the selected Zone Budget zone.

The checkboxes beside each term can be used to add or remove the individual source and sink terms from either the Time Series graph or the Time step graph.

The group checkboxes will add or remove the entire IN/OUT group.

The Percent Discrepancy graph and the IN-OUT graph consider only the total flow into and out of the entire system as a whole. As a result, the settings for the individual IN and OUT terms do not influence the results and appearance of these graphs.

Note: By convention, Zone Budget considers mass balance from the perspective of the groundwater system. For example, the IN term for River Leakage represents the river leakage INTO the groundwater system.

Arranging the Graphs

Each of the graphs can be maximized to occupy the entire Zone Budget window by clicking the maximize button in the top right-hand corner of each graph window. Alternately, the graphs can be tiled in the Zone Budget window by selecting Window/Tile from the top menu bar.
10.12.3 Compare Heads and Drawdown

Visual MODFLOW Flex allows you to compare Heads or Drawdown between several model runs. This is useful to determining differences in pre and post pumping heads or comparing impacts to heads or drawdown when adjusting property or boundary condition parameters.

Follow the steps below:

- Right-click on the Heads node of any model run, and select the "Compare" option. The following dialog will appear

There are two options available:
- Select a Heads from the Output tree from another model run; in this case, click on the Heads node from another model run, and click on the button to add this to the field
- Select a MODFLOW .HDS file from the hard drive, from another model. Click on the Open folder button and browse to a .HDS file on the hard drive.
- Click OK. A new view will appear. The plot shows the results of subtracting the heads selected specified above from the Heads where you launched the `Compare Heads` option.
The same option is available for Drawdown. Just launch `Compare` from the Drawdown node on the Output model tree, and choose the Drawdown from another model run, or a .DDN file from the hard disk.

**Limitations:**
The comparison calculation has the following assumptions: Both models must have the same:

- start date (as defined in the Conceptual Model settings),
- stress periods,
- spatial coverage (XY extents), and
- angle of rotation

### 10.13 Duplicating Models

Traditionally, multiple scenarios of a model would be handled by saving the entire project as a new file. This makes it difficult to keep track of various versions of the model and which inputs lead to specific results, etc.

In Visual MODFLOW Flex, a numerical model run can be duplicated or 'cloned' and managed and modified in a single project. This facilitates an iterative approach to modeling by allowing
you to adjust a few input parameters, translate and run, and interpret the results. You can also Compare Heads or Drawdown between model runs to other scenarios.

To Clone a numerical model, follow the steps below.

After this the model tree will be duplicated and added under the selected numerical grid; a new Numerical Model workflow will then load where you can work with this model run clone.
The Workflow and Model Explorer are closely associated; as you progress through a specific numerical model workflow, the items you see in the Flex Viewer originate from the corresponding Run folder on the Model Explorer. For example, if you are working a clone of a model run, eg. Run2, and you are viewing boundary condition cells, you will see these originate from the After this the model tree will be duplicated and added under the selected numerical grid; a new Numerical Model workflow will then load where you can work with this model run clone.

### 10.14 Exporting Data

Visual MODFLOW Flex allows you to export some raw data and most of the numerical model inputs and outputs to shapefiles for external analysis.
**Data Objects**

Data objects in the Data Explorer can be exported to external formats as follows:

- Points, Polygons and Polylines can be exported to shapefiles (*.SHP), or comma-separated values text files (*.CSV)
- Surfaces and Horizons exported to comma-separated values text files (*.CSV)
- Surfaces that have been generated based on a uniform grid spacing (i.e. \( \Delta X = \Delta Y = \) constant) can also be exported to a Surfer ASCII grid files (*.GRD)

To export a data object:

- From the Data Explorer, right-click on the desired data object, and select Export from the pop-up menu.
- A Save As dialog box will display on your screen
- Specify a file Name and Folder location, and the File Type for the exported file, and then click the [Save] button.
- Click the [OK] button from the Export dialog box.

**Model Explorer Objects**

**Exporting to Shapefiles**

Export to points or polygon shapefile is currently available for the following:

- **Model Inputs:**
  - Numerical Grid
  - Properties: Conductivity, Storativity, Initial Heads, Longitudinal Dispersion, Species Parameters, Initial Concentrations, and Bulk Density
  - Boundary Conditions: Recharge, Evapotranspiration, and UZF
  - Zone Budget Zones

- **Model Outputs:**
  - Heads
  - Drawdown
  - Budget
  - Velocity
  - Concentration: each constituent species is exported separately

When exporting point shapefiles, one point will be exported for each applicable cell centroid. When exporting polygon shapefiles, one polygon will be exported for each applicable cell.

To export data on the model explorer, right-click on the desired item (eg. Conductivity), and select Export.... as shown below.
The following dialog will appear; note at the bottom you have the option to choose Points Shapefile or Polygon Shapefile depending on the type of data to be exported.

Click OK. Depending on the type of data, you may have additional settings. The following options are available for Properties:
Choose which attributes you want to be generated and what layers should be included.

Click OK and the shapefile will be created.

**Exporting Output Heads and Concentrations**

The following additional options are available for exported output Heads and Concentrations:
Choose which attributes you want to be generated and what layers should be included.

If your model has transient heads, you have an option to include Heads from multiple times as separate attributes in the shapefile .DBF file. Choose specific times or 'Select Heads at All Times' to include all heads.
### Notation in Shapefile DBFs

There are some restrictions on attribute names in DBF files; the length is restricted to 10 characters, and certain characters cannot be used, including decimal (.), hyphens (-), and plus sign (+). As such, Visual MODFLOW Flex uses the following notation:

For transient heads, the attribute name notation is HT (head at time) followed by the time value - an underscore is used in place of a decimal, M is used in place of hyphen, and is used in place of plus.

For example:

<table>
<thead>
<tr>
<th>Calculated Notation in head times DBF file</th>
</tr>
</thead>
<tbody>
<tr>
<td>1E-5</td>
</tr>
<tr>
<td>500.25</td>
</tr>
<tr>
<td>9E+7</td>
</tr>
</tbody>
</table>

### Exporting Output Budget

You are also able to export flow budget data for the model and Boundary Condition cells as a comma separated variable (.CSV) text file. After you have run your model you will find Budget in the Output of the model explorer - right click and select export and the Export fluxes dialog will appear.
You must specify the location of the file. You can also select which layer(s), time step(s), and attributes(s) to export. The XRate, YRate, ZRate, InflowRate, OutFlowRate, and NetFlowRate will always be available in the attributes list. While the attributes list will also contains a list of boundary conditions budget terms that are active in the model (in the case shown above the list includes Wells, Drains, Recharge, Constant Heads, and Rivers).

**Exporting Contours**

Output contour lines can be exported as polyline shapefiles for further analysis and mapping in external applications. To do this, enable the 3D subview to render the contours, then right-click on the contoured output (e.g. Heads, Drawdown, Concentrations) and select "Export contours...". A polyline shapefile will be created for the contours as specified in the relevant object settings. You can adjust the contour lines, including adding custom lines, using the `ThreeSlices>ContourLine` settings in the 3D subview or the `ContourLine` settings in a 2D subview.

**Exporting Particle Pathlines**

Pathline and time markers can be exported to shapefiles for further analysis; right-click on the Forward or Backward Pathlines on the model explorer, and choose Export. A points shapefile will be created for the time markers, and a polyline shapefile will be created for the pathlines. The exported files will utilize the pathline options you have currently defined in the view for the tracking time and time marker frequency. See the section on visualizing particle track results for more information.
Exporting to Text Files

Export to Text-IJK/.CSV files is currently available for the following:

- **Model Inputs**
  - Model Layers
  - Boundary Conditions: all, including wells, but excluding recharge, evapotranspiration, and UZF

- **Model Outputs**
  - Water Table

You can export your Model Layers to .CSV or .TXT by right-clicking on the grid in the Model Explorer and selecting Export Layers...

Then you can provide the location and select if you want .CSV or .TXT.

Once you select Save you will be notified that the layers have been exported and given the option to open the exported file.

The grid layers have been exported successfully. Do you want to open the file?

Yes  No

The resulting file will contain Layer, Row, Column, X, Y, Z, Top, Bottom, and Thickness information.
Boundary conditions can be exported using the same steps described for exporting Model Layers above. Boundary condition data fields will be exported along with the Layer, Row, and Column data fields. Data fields for stress period start and end times will be added if the boundary condition is transient.
11 Numerical Modeling Workflow - Unstructured Grids (MODFLOW-USG)

This chapter presents information on editing data in a numerical model workflow; the numerical model for MODFLOW-USG is generated after you run the Conceptual to Numerical conversion step.

**Note**

You must have a license of Pro or Premium in order to use MODFLOW-USG in Visual MODFLOW Flex.

The numerical modeling workflow provides the tools for defining, viewing and editing the numerical model (properties and boundaries assigned to grid cells), creating the input files for MODFLOW-USG, running the MODFLOW-USG engine, and analyzing the results. The following sections are covered:

1. Define Modeling Objectives
2. Define Properties
   - (edit parameter values, display in plan, XS, and 3D views)
3. Define Boundary Conditions
   - (constant heads, drains, pumping wells, recharge, etc.)
4. Select Next Step
   a. Define Observation Wells
   b. Define Zone Budget Zones
   c. Define Particles
   d. Select Run Type/Single Run
5. Translate
6. Run
7. View Results

For more details on how MODFLOW-USG differs from traditional versions of MODFLOW, please see our website.
11.1 Define Properties

This step allows you to adjust the display and rendering of the USG cells, and also edit the attributes of individual cells or zones.

For a saturated groundwater flow model, there are three parameter groups: Conductivity, Storage, and Initial Heads. By default, hydraulic conductivity will be displayed, and rendered by the ZoneID. In order to see another parameter group, you need to turn off (remove the check box beside) Conductivity from the tree, and turn on (place a check box beside) the new desired parameter group (e.g. Initial Heads).

Assign

Property zone values can be assigned directly to selected numerical cells using the Assign options from the toolbox.

- **Single**: Select one or more cells with the left mouse button; a small red point will appear over the selected cell
- **Polyline**: Draw one or more polylines to select the desired cells
- **Polygon**: Draw one or more polygons to select the desired cells
- **Entire Layer/Row/Column**: Selects the currently selected layer/row/column (all active cells)
• **Using Data Object:** Use data object (polygon or polyline) to select the desired cells

**Edit**

Select the Edit button to edit Property zone values after they have been assigned and the Edit property dialog is launched. Here you can select the zone and then you will have the ability to edit any of the cells that are part of the zone.

![Edit property dialog](image)

The following buttons are available to assist with your edits:

- Assign to current cell
- Assign to column
- Apply to selected cells

Also available is the Script tab. If preferred you can make your edits by using the Scrip option. On this tab you will find the script for any edits you make on the Edit Cells tab allowing you to learn the script language. You can then use the script to "replay" the same edits again.

**Display Settings**

Most of the display settings for a 3D UnStructured grid for properties are the same as what you have for a 3D Structured (Finite Difference) grid; these are explained in the settings section, see [3D Gridded Data](#).

The following is a list of the differences for MODFLOW-USG grids

- Under Cells, there is no option for "Show Only Borders"
- For Lines, there is no option for "Show Only Grid Frame"
- For Slice, there is an option "Show Cell ID Labels"; this will insert the MODFLOW-USG CellID for each cell, for the selected slice. This will be useful if you want to adjust the numerical values for single cells (or a group of cells) using the Edit option described below.
For Slice and ThreeSlices, the slice position for cross-sectional slices is based on a virtual grid; the UnStructured grid is divided into 100 equally spaced rows and columns. You can display cross-sections through the USG, as "rows" or "columns", at positions from 1 - 100. Row 1 corresponds to the Y minimum in your model domain, whereas Row 100 corresponds to the Y maximum in your model domain. Column 1 corresponds to the X minimum in your model domain, whereas Column 100 corresponds to the X maximum in your model domain. The layer convention follows the MODFLOW notation, where Layer 1 is at the top.

For Colormap, the positions are similar as to what you have for Slice described above. The option to display colormap along an arbitrary Cross-section line is currently not supported. For Colormap, there is an additional Slice Type, "Horizontal", which allows for rendering on a flat horizontal plane. In this case, you define the vertical position (shift) of this Horizontal plane, on a location from 1 - 100. Vertical Shift of 1 corresponds to the Z minimum in your model domain, whereas a shift of 100 corresponds to the Z maximum in your model domain. Only active cells in the model domain will be drawn.

For Isolines (contour lines), the settings for selecting the plane for drawing the isolines are similar to described for Colormap above.

**Export**

Any of the model properties can be exported to shapefile for further post-processing; see Export for more details.

### 11.2 Define Boundary Conditions

With MODFLOW-USG boundary conditions are still defined at the cell-center, as with structured versions of MODFLOW. One of the biggest advantages of UnStructured grids is that the cell sizes can be much smaller around these boundary conditions, providing for a higher level of accuracy and better representation.

The other major differences lie in the way boundary condition cells are identified. With Structured MODFLOW (-2005, etc.), boundary condition cells are identified by row, column, layer, in their respective packages (.RIV, .WEL, .GHB, etc.). With MODFLOW-USG, cells are identified by a CellID; after that, the parameters remain the same as what you see for Structured MODFLOW.
The CellID label for boundary condition cells can be turned on as described below.

**Assign**

The Assign button allows you to assign new boundary conditions. You have options to assign by Polyline or Polygon, by Using Data Object or by Cells. These options are described further below.

- **Polygon or Polyline**: Use this option to digitize one or more shapes and assign Boundary Condition attributes to the respective shapes. Use the left mouse button to start drawing and add attributes at the desired locations. Once you have finished digitizing, right click with the mouse to close the shape and select Finish button.

- **Using Data Object**: Use this option when you have a polyline or polygon data object already imported into the projects, and you would like to use this shape to define the geometry.

- **Cells**: Use this option when you want to select individual cells to assign a boundary condition.

The Edit Boundary Condition attributes window will appear as shown below:
Enter the attributes using the various methods; click Finish when you are done. This dialog allows you to copy and paste from Excel to make defining (or editing) the attributes easy.

For more details on the options for defining attributes, see Define Boundary Conditions (Attributes).

If your boundary condition is transient switch the schedule drop down to transient to enter your schedule.

You can copy and paste your time schedule from Excel into this dialog to make defining your boundary condition quicker and easier.

By default the Assign to appropriate layer is selected. This means if you enter values for the attributes of a boundary condition (for example the Starting Head) which would place it into another layer - this will occur automatically once you save your boundary condition.

**Edit**

When you select the Edit... button you must select a cell containing the boundary condition you wish to edit.

Then the following dialog will appear (similar to defining the boundary condition):
The attributes for each cell containing the boundary condition are displayed and can be edited. The cell you selected will be highlighted in the dialog. If you select other rows in the dialog - the corresponding cell will be highlighted in the viewer.

By default the Assign to appropriate layer is selected. This means if you adjust the attributes of a boundary condition (for example the Stage) which would place it into another layer - this will occur automatically once you save your edits.

**Script**

If preferred you can make your boundary condition edits by using the Script option. On this tab you will find the script for any edits you make on the Edit cells tab allowing you to learn the script language. You can re-use the script to “replay” your edits to your boundary conditions.

**Display Settings**

The display settings can be launched by right-clicking on the desired object in the model tree, and selecting Settings...

Boundary condition cells can be shown:

The Setting window will load.

Boundary Condition cells can be drawn as Points (vertices) or following the cell geometry (Cells)
When selecting the Cells option, you will see the option "Show Cell ID labels"; turning this on will draw the Cell ID on top of the selected boundary condition cell type. This is useful if you wish to make edits to specific boundary condition attributes, as explained in the following section.

To delete a specific boundary condition cell group, right click on this node in the tree, and select Delete.

To validate there are no warning or errors associated with a boundary condition, right click a specific boundary condition and select Validate... and if there are any errors or warning you will be provided an option to resolve the issue as shown in the dialog below. This is available for Constant Head, River, General Head, and Drain. It will check to see if the attributes of the boundary condition will place it below the bottom of the cell you assigned it to. Then it will give you options to correct this. If you choose to Ignore the errors you may encounter errors when trying to translate and run your model.

![Resolve validation failures dialog](image)

11.3 Define Observations

Observation wells can be added to the numerical model as follows.

First, import the observation well data as described in the Import Wells section.

The observation well data can be viewed/edited in the Data Table section.

Once you have the head observation data defined, navigate to the Define Observations step in the USG numerical model workflow.
Above the 3D Viewer, you will see the insert blue arrow; select your Head Observation Wells data object from the data explorer and click on the button. The observation wells should then appear in the 3D view as green points, and will also appear as a new node on the numerical model tree.
The vertical position of the head observation will be determined by the Elevation value of the Head Observation point defined in the Wells Data Table.

**Observation Groups**

If you assign subsequent Observation Wells data objects from the data explorer will be prompted with a choice to append the observations to an existing well group or create a new well group.

![Select Well Group](image)

**View / Edit in CSV Format**

The head and concentration observation data in an observation data group can be viewed/modified in a CSV format (once it has been created).

- Right-click on a Head (or Concentration) Observations Group node from the Model explorer
- Select "Edit Attributes..." as shown below.
The data contained inside the Observations data object will be displayed in a .CSV file; this should appear in either Notepad or Excel depending on your system configuration; an example is below:
You can use this interface for numerous types of edits/additions:

- Adding a new observation point at a defined XY location
- Change existing data
- Add a new observation time to an existing observation point.

**Please Note:** Note that the Layer number for the observation point is calculated based on the Obs. Elev; changing the layer number will not change the vertical location of the observation point).

When you are finished with the edits in the .CSV file, you need to save and close this file. Then re-import back the changes into Visual MODFLOW Flex using the steps below:

- Right-click on the Head (or Concentration) Observations node from the Model explorer
- Select “Reload Attributes...”
The modified data will now appear and will be utilized for any charts the next time you complete a translate and run of the model.

11.4 Define Zone Budget Zones - USG

ZoneBudget-USG is a USGS program for computing subregional water budgets for MODFLOW ground-water flow models using unstructured grids. Within Visual MODFLOW Flex, you can create new zones for MODFLOW-USG models in the numerical model workflows. At the Define Zone Budget Zones step, you can create new zone budget zones by digitizing new zones using Assign Polyline, Polygon, or Point (Single Cell), or by using 2D/3D point, polyline, or polygon shape objects from the Data tree using the same methods as in the Finite Difference workflow.

Follow the steps below to create new zone budget zones:

**Assign**

**Polyline**
- Use the left mouse button to start digitizing a polyline over the desired cells. Once you have finished digitizing, click on the Finish button from the toolbox, or right click with the mouse and select "Define Attributes". The following window will appear
Click the New button to assign the selected cells to a New zone; otherwise, you can select to assign cells to an existing zone by selecting the zone from the dropdown list at the top. The selected cells can be assigned to the current layer or multiple layers using the check boxes at the bottom of the window. Once you are finished, click OK.

**Polygon**
- Digitize one or more polygons, and repeat the steps as described above for Polylines.

**Single**
- Left-click on one or more cells and a point will appear. Once finished, right-click and select "Define Attributes", and follow the steps as described above for Polylines.

**Using Data Object**
- The following window will appear.
  - Choose the desired point, polyline, or polygon data object from the tree.
  - Click on the button (under Select Data Object)
A second window will appear:

- Select which zone you want the features to apply to. You have the option to assign the zones to:
  - An existing zone
  - A new single zone
  - Create zones for individual features
  - Create zones based on feature attributes
- Select the desired model layers where the zone should be applied

A simple example is shown below for a polygon data object (shown in blue).
After the zones are generated, they will appear in layer view.

When the selected shape contains multiple polyline or polygon features, then a new zone will be created for each feature.
11.5 Define Particles

Particles are defined using the same methods as in the Finite Difference workflow. These methods are described in the section describing how to Define Particles in the Finite Difference workflow.

11.6 Select Engines

The Single Run workflow step is used to select the Flow and Transport Engines you wish to include in the model run. Visual MODFLOW Flex requires you to select a flow engine for a particular run. Running a particle track engine and/or ZONEBUDGET-USG is typically optional.
The table below lists all supported engines and utilities supported in Visual MODFLOW Flex for unstructured grids.

**Engines supported in the Numerical Workflow**

<table>
<thead>
<tr>
<th>Flow</th>
<th>Transport</th>
<th>Utilities</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODFLOW-USG</td>
<td>N/A</td>
<td>ZONEBUDGET-USG and MOD-PATH3DU</td>
</tr>
</tbody>
</table>

When you are finished, click (Next Step) button to proceed.
11.7 Translation Settings

The Translation step allows you to translate the model into the input files that the applicable engine(s) will run. There are also settings for each engine that allow you to make adjustments to how the model is run (e.g. solver settings, initial heads, output control).

The translation step consists of two vertical panels. The left panel includes a tree structure that lists the engine(s) selected at the Select Engines step and their options that have settings:

- **MODFLOW-USG**
  - Solvers
  - Initial Heads
  - Output Control
  - Advanced Settings

- **MOD-PATH3DU**

**General Settings**

At the General Settings page you can see the output directory where the translated files will be generated. Note that you can access the directory by clicking the folder icon (📁).

You can also see the start date. The start date will be the same as that which you specified in the Define Modeling Objectives step.

The [Translate] button will translate the model input data from Visual MODFLOW Flex file formats to the data files required for the selected numeric engines (see Appendix A).

Upon translation of your model, a series of tabs will be generated for each translated package. You can click any of these packages to view the contents of the corresponding input file. You can also edit these and save your changes by right-clicking in the package.
window and selecting save or by typing \texttt{CTRL S}. Note that your changes will be overwritten if you re-translate your model and the package advanced \texttt{setting} for translation is set to"Yes". Tabs can be displayed along a single line with scrolling arrows or along multiple lines. To toggle this behavior, click the multi-line button ( ).

11.7.1 MODFLOW-USG

The adjustable settings available for the MODFLOW-USG engine are described below.

MODFLOW-USG Settings

**Property Package**: available options are:
- Layer Property Flow (LPF) (default) or
- Block-Centered Flow (BCF)

**Run Type**: Steady-State or Transient

If the Steady-State Flow option is selected, Visual MODFLOW Flex will prepare the data set for a steady-state flow simulation, and will automatically use the data from the first time period (only) of each boundary condition and pumping well defined in Visual MODFLOW Flex to run the model to achieve flow equilibrium (i.e. a time-independent solution since all inputs are constant).

Please Note: if running a steady-state model, at least one boundary condition with a head of known elevation must be specified (e.g. constant head, river stage, drain bottom, general head elevation).

If the Transient Flow option is selected, Visual MODFLOW Flex will prepare the data set for a transient flow simulation. During this process, Visual MODFLOW Flex will automatically merge all of the different time period data defined for each pumping well and boundary condition into the stress period format required by the applicable engine. This creates a time-dependent flow solution, as the model is being run with different inputs at different times.

**Steady-State Simulation Time**: specified time for the mass balance terms in a steady-state run.
Stress Periods and Time Steps

The Time Steps option is only available when you are running a transient model (i.e. when Transient Flow run type is selected). For transient flow simulations, Visual MODFLOW Flex will automatically merge all of the different time periods defined for all of the different pumping wells and boundary conditions into the uniform stress period format required by MODFLOW. A stress period is defined as a time period in which all the stresses (boundary conditions, pumping rates, etc.) on the system are constant. Unfortunately, the data collected for each modeling site is rarely synchronized in terms of stress periods, so Visual MODFLOW Flex merges the time schedules for all pumping wells and boundary conditions to determine the length of each stress period for a transient simulation. As a result, the user cannot directly modify the number of stress periods or the length of each stress period.

The Time step options window (as shown in the following figure) is used to define the number of Time steps in each stress period and the time step Multiplier is used to increment each time step size.

The Stress Period column indicates the stress period number while the Start and End columns indicate the start time and end time, respectively, for each stress period.
Each stress period is divided into a user-defined number of Time steps whereby the model will calculate the head solution at each time step. The default value for Time steps is 10.

The time step Multiplier is the factor used to increment the time step size within each stress period (i.e. it is the ratio of the value of each time step to that of the preceding time step). The default value is 1.2. A time step Multiplier value greater than 1 will produce smaller time steps at the beginning of a stress period resulting in a better representation of the changes of the transient flow field. Thus increasing the number of time steps in a simulation may result in smoother head or drawdown versus time curves.

The steady-state column indicates if the stress period is transient or steady-state. Steady-state and transient stress periods can occur in any order. Commonly the first stress period may be run as steady state, to produce a solution that is used as the initial condition for subsequent transient stress periods.

Accommodating Boundary Conditions with Varying End Times
Visual MODFLOW Flex allows you to define boundary conditions with varying end times; the number of stress periods for the model will be calculated from the end time of the longest boundary condition. If there is a stress period where there are no data defined for a boundary condition, then you will see an entry of "0" in the appropriate package, for that stress period; this denotes that zero cells are defined for that stress period. In the case of pumping wells, you will see a rate of 0 for the well cells.

The Model Run Type is either Steady-State or Transient
The settings for some of the other packages can be adjusted as per the following sections.
For more details, please refer to the MODFLOW-USG version 1.4.00 manual, by Panday et al. (2013), which is available from the USGS website: http://water.usgs.gov/ogw/mfusg/

Layer Types
By default, the layer type is set as 4, for both the .BCF and .LPF packages. The layer types can be adjusted at the top of each respective package.

Time Steps
The number of time steps for each stress period is defined at the end of the .DISU package; simply open this tab, change the desired value, save the package, then proceed with the run.

Output Control
The OC package for MODFLOW-USG uses text keywords, making it simpler to understand the file format. By default, heads and drawdown will be saved for every time step, up to 10
time steps per stress period. If you change the number of time steps, then you may need to update the Output control, to save heads/drawdowns at the desired time steps.

**SMS Package**
The parameters for the .SMS package are described in the MODFLOW-USG manual, and in the "MODFLOW-USG Version 1.4: Description of Model Input and Output". In addition to the options described in the manual, you can also choose to use the SAMG solver for the matrix solution method (i.e. LINMETH=3).

**Initial Heads**
You can select

Initial Head values are saved in the .BAS package; if you want to use a Heads file from a previous MODFLOW-USG run, you can update the .BAS file with the link by following steps 1 through 8 below:

1. Run MODFLOW-USG to create the desired .HDS file
2. Open this model Run directory (in Windows Explorer), and make a copy of this file, saved with a new name (such as model-initial-heads.HDS)
3. Go back to the Translate step.
4. Select the .BAS tab
5. Browse to about the half-way point in the file, looking for the section:

   "INTERNAL       1.0 (10F15.0)   -1 Initial Head layer Layer 1"

6. Delete this line, and all rows after this line (containing the default initial head arrays).
7. Insert the following line:

   OPEN/CLOSE 'D:\project_directory\project.data\MODFLOW\UnstructuredGrid2\Run1\MODFLOW-USG\initial-heads.HDS' 1 (BINARY) -1

   Where the path and filename (shown in blue) match the precise location of the .HDS file from the previous MODFLOW-USG run that you wish to use.

8. Save the .BAS file, then Run

**BAS Package and Discontinuous Layers**
Visual MODFLOW Flex assumes that all layers have the same horizontal discretization. When there is a layer that pinches out, resulting in a cell thickness of zero, these cells will not be displayed in the 3D viewer. During the translation to MODFLOW-USG packages, the cells with zero thickness will be marked as inactive in the basic (.BAS) package and in the .DISU (UnStructured Grid) package, those zero thickness inactive cells have no vertical connections. Their upper and lower neighbors are directly connected together, effectively bypassing the inactive cells with zero-thickness.
11.7.1.1 Solvers

Solvers in MODFLOW-USG are handled through the Sparse Matrix Solver (SMS) package. The SMS package uses a combination of strategies including: preconditioning, acceleration, linear and non-linear methods, ordering, and term dropping to solve the simultaneous equations produced by the model as described below.

General Solution Method Parameters

The following parameters and settings are always incorporated in the SMS package.

Pre-defined input values (OPTIONS)

An optional keyword setting that activates solver options:

- **SIMPLE** indicates that default solver input values will be defined that work well for nearly linear models. This would be used for models that do not include nonlinear stress packages (e.g. UZF, SFR) and models that are either confined or consist of a single unconfined layer that is thick enough to contain the water table within a single layer.

- **MODERATE** indicates that default solver input values will be defined that work well for moderately nonlinear models. This would be used for models that include nonlinear stress packages and models that consist of one or more unconfined layers. The “MODERATE” option should be used when the “SIMPLE” option does not result in successful convergence.

- **COMPLEX** indicates that default solver input values will be defined that work well for highly nonlinear models. This would be used for models that include nonlinear stress packages and models that consist of one or more unconfined layers representing complex geology and sw/gw interaction. The “COMPLEX” option should be used when the “MODERATE” option does not result in successful convergence.

- **CUSTOM** indicates that no keyword will be included and the full list of applicable settings will be translated into the SMS package.

⚠️ Please Note: when using the SIMPLE, MODERATE, and COMPLEX options, only the following parameters are included in the SMS package: HCLOSE, HICLOSE, MXITER, ITER1, IPRSMS, NONMETH, LINMETH. The defaults are used for the remaining settings.

Head Change Criterion (HCLOSE)

The head change convergence criterion for nonlinear problems. HCLOSE can be any positive value; the default is 1.0E-5.
Head Change Criterion for Convergence (HICLOSE)
The head change convergence criterion for inner iterations. HICLOSE can be any positive value; the default is 1.0E-5.

Max. Outer Iteration (MXITER)
The maximum number of outer iterations. MXITER can be any positive integer value; the default value is 500.

Max. Inner Iteration (ITER1)
The maximum number of inner iterations for each linear solution. ITER1 can be any positive integer value; the default value is 1600.

Printing Convergence Info (IPRSMS)
Controls printing of convergence information from the solver in the Listing file.
   - Don't display convergence information (0): excludes solver information from the listing file
   - Display total iterations and summaries (1): includes the total number of inner/outer iterations and the nonlinear residual reduction summaries [default]
   - Display all available information (2): includes the information above, plus matrix solver information

Nonlinear Solution method (NONLINMETH)
Controls the nonlinear solution method and under-relaxation schemes. The absolute value of NONLINMETH determines the underrelaxation scheme used, while the sign determines the iteration scheme:
   - 0: Picard iteration scheme is used without any under-relaxation schemes involved
   - < 0: Picard iteration scheme is used with under-relaxation.
   - > 0: Newton-Raphson iteration scheme is used with under-relaxation.
   - 1 or -1: Delta-Bar-Delta under-relaxation is used.
   - 2 or -2: Cooley under-relaxation scheme is used.

The default is 2 which corresponds to Newton-Raphson iteration with Cooley under-relaxation

Linear Solution Method (LINMETH)
The SMS package includes the option to use one of three solvers:
- Use Ibaraki chi-MD matrix solver (LINMETH=1): an asymmetric matrix solver called chi-MD (Ibaraki, 2005),
- Use White-Hughes matrix solver (LINMETH=2): an unstructured preconditioned conjugate gradient (PCGU) solver developed by White and Hughes (2011) for symmetric equations, and
- Use SAMG matrix solver (LINMETH=3): the advanced/proprietary SAMG solver developed at the Frauenhofer Institute.

Non-linear Solver Parameters

The delta-bar-delta algorithm is a pre-conditioning method that is based on neural networks that uses results from previous iterations to inform the direction and magnitude of cell-by-cell under-relaxation. The Cooley scheme is a less-memory intensive alternative to the delta-bar-delta scheme.

The following parameters complement the gradient-based solutions and define how much and what components of the previous results are used to inform the overall search gradient relative to the search gradient determined by the linear solution method alone:

**Reduction Factor for Learning Rate of Delta-Bar-Delta algorithm (THETA)**

If the change in the head is of opposite sign to that of the previous iteration, the under-relaxation term is reduced by a factor of THETA. THETA can be any value from 0 to 1 and usually ranges from 0.3 to 0.9. The default value is 1.0 but a value of 0.7 works well for most problems.

**Learning Rate Increment (AKAPPA)**

The increment for the learning rate (under-relaxation term) of the delta-bar-delta algorithm. If the change in the head is of the same sign to that of the previous iteration, the under-relaxation term is increased by an increment of AKAPPA. AKAPPA can be any value from 0 to 1 and usually ranges from 0.03 to 0.3. The default is value is 0 but a value of 0.1 works well for most problems.

**Memory term factor (GAMMA)**

The history or memory term factor of the delta-bar-delta algorithm. The memory term is maintained as an exponential average of past changes. Retaining some past history can overcome granular behavior in the calculated function surface and therefore helps to overcome cyclic patterns of non-convergence. When GAMMA is zero, only the most recent history (previous iteration value) is maintained. As GAMMA is increased, past history of iteration changes has greater influence on the memory term. GAMMA can be any value from 0 to less than 1 and usually ranges from 0.1 to 0.3. The default value is 0.1, but a value of 0.2 works well for most problems.
**Fraction of past history added (AMOMENTUM)**

The fraction of past history changes that is added as a momentum term to the step change for a nonlinear iteration. A large momentum term should only be used when small learning rates are expected. Small amounts of the momentum term help convergence. AMOMENTUM can be any value from 0 to 1 and usually ranges from 0.0001 to 0.1. The default value is 0, but a value of 0.001 works well for most problems.

**Backtracking Iterations Allowed (NUMTRACK)**

The maximum number of backtracking iterations allowed for residual reduction computations. If NUMTRACK = 0 then the backtracking iterations are omitted. NUMTRACK can be any non-negative integer and usually ranges from 2 to 20. The default is 0 and a value of 10 works well for most problems.

**Residual Change Tolerance (BTOL)**

The tolerance for residual change that is allowed for residual reduction computations. BTOL should not be less than one to avoid getting stuck in local minima. A large value serves to check for extreme residual increases, while a low value serves to control step size more severely. The value usually ranges from 1.0 to 106; a value of 104 works well for most problems but lower values like 1.1 may be required for harder problems.

**Step Size Reduction (BREDUC)**

The reduction in step size used for residual reduction computations. BREDUC can be any value from 0 to 1 and usually ranges from 0.1 to 0.3. The default value is 0.1 and a value of 0.2 works well for most problems.

**Residual reduction limit when backtracking (RESLIM)**

The limit to which the residual is reduced with backtracking. If the residual is smaller than RESLIM, then further backtracking is not performed. A value of 100 is suitable for large problems and residual reduction to smaller values may only slow down computations.

**Parameters for the Ibaraki MD Solver**

The following parameters are used by the Ibaraki MD Solver:

**Acceleration method flag (IACL)**

The flag for choosing the acceleration method.

- Conjugate Gradient (0): select this option if the matrix is symmetric. [Default]
- ORTHOMIN (1)
• BiCGSTAB (2)

Ordering Scheme Flag (NORDER)
The flag for choosing the ordering scheme.
• Original ordering (0) [Default]
• Reverse Cuthill McKee ordering (1)
• Minimum degree ordering (2)

Fill Level for ILU Decomposition (LEVEL)
The level of fill for ILU decomposition. Higher levels of fill provide more robustness but also require more memory. For optimal performance, it is suggested that a large level of fill be applied (7 or 8) with use of drop tolerance. The default value is 3.

Number of Orthogonalizations (NORTH)
The number of orthogonalizations for the ORTHOMIN acceleration scheme. A number between 4 and 10 is appropriate. Small values require less storage but more iteration may be required. This number should equal 2 for the other acceleration methods.

Reduced System Index (IREDSYS)
The index for creating a reduced system of equations using the red-black ordering scheme:
• No reduced system of equations (0) [Default]
• Create reduced system using red-black ordering (1)

Residual Tolerance Criterion (RRCTOL)
A residual tolerance criterion for convergence. The root mean squared residual of the matrix solution is evaluated against this number to determine convergence. The solver assumes convergence if either HICLOSE (the absolute head tolerance value for the solver) or RRCTOL is achieved. Note that a value of zero ignores residual tolerance in favor of the absolute tolerance (HICLOSE) for closure of the matrix solver.

Drop Tolerance (IDROPTOL)
The flag to perform drop tolerance:
• 0 – do not perform drop tolerance
• 1 – perform drop tolerance [Default]
Drop Tolerance Value (EPSRN)
The drop tolerance value. A value of $10^{-3}$ works well for most problems.

Parameters for the Hughes-White PCGU Solver
The following parameters are used by the Hughes-White PCGU Solver:

Linear Acceleration Method (CLIN)
An option keyword that defines the linear acceleration method used by the PCGU solver.

- Preconditioned conjugate gradient method (CG) [Default]
- Preconditioned bi-conjugate gradient stabilized method (BCGS)

The preconditioned conjugate gradient method should be used for problems with a symmetric coefficient matrix. The preconditioned biconjugate gradient stabilized method should be used for problems with a non-symmetric coefficient matrix (for example, with problems using the Newton-Raphson linearization scheme).

Pre-Conditioner Flag (IPC)
A flag for choosing a matrix preconditioner:

- No preconditioning (0)
- Jacobi preconditioning (1)
- ILU(0) preconditioning (2)
- MILU(0) preconditioning (3) [Default]

Matrix Scaling Approach Flag (ISCL)
A flag for choosing the matrix scaling approach used:

- No matrix scaling applied (0) [Default]
- Symmetric matrix scaling (L-squared norm) (2): use the $l_2$ norm of each row of $A$ (DR) and the $l_2$ norm of each row of $DRA$.

If the ILU(0) or MILU(0) preconditioners are used (IPC = 2 or 3) and matrix reordering is selected (IORD > 0), then ISCL must be 1 or 2.
Matrix Reordering Approach Flag (IORD)

A flag for choosing the matrix reordering approach used:

- Original ordering (0) [Default]
- Reverse Cuthill McKee ordering (1)
- Minimum degree ordering (2)

If reordering is used, reverse Cuthill-McKee ordering has been found to be a more effective reordering approach for the test problems evaluated.

Flow Residual Tolerance for Convergence (RCLOSEPCGU)

A real value that defines the flow residual tolerance for convergence of the PCGU linear solver. This value represents the maximum allowable residual at any single node. Value is in units of length cubed per time, and must be consistent with MODFLOW-USG length and time units. Usually a value of 0.1 is sufficient for the flow-residual criteria when meters and seconds are the defined MODFLOW-USG length and time units.

Relaxation Factor (RELAXPCGU)

A real value that defines the relaxation factor used by the MILU(0) preconditioner. RELAXPCGU is unitless and should be greater than or equal to 0.0 and less than or equal to 1.0. RELAXPCGU values of about 1.0 are commonly used, and experience suggests that convergence can be optimized in some cases with RELAXPCGU values of 0.97. A RELAXPCGU value of 0.0 will result in ILU(0) preconditioning.

RELAXPCGU is only specified if IPC=3. If RELAXPCGU is not specified and IPC=3, then a default value of 0.97 will be assigned to RELAXPCGU.

11.7.1.2 Initial Heads

In Visual MODFLOW Flex, the Initial Heads are defined at the stage of Defining the Properties. For more details, please see the section on setting Initial Heads in the Define Properties workflow step.

When translating your model you have the following settings for Initial Head:
- Use Specified Heads - will use the values that are defined for “Initial Heads” at the Define Properties step.
- Use Ground Elevation - will use the elevation values for the top of Layer 1.
- Use Previous MODFLOW Run - requires you to select a .HDS file from a previous MODFLOW Run and specify the Time step to be used.

**Warning!**
Using Heads from Previous MODFLOW Run
The selected .HDS file cannot be the same as the .HDS file in the current translation directory. If you select Use .HDS from Previous MODFLOW Run option, you must choose a .HDS file from another directory.
Output Control

The Output Control run options set the information and frequency of information written and saved to the various MODFLOW-USG output files (see following figure).

Each MODFLOW simulation can produce three binary output files and one ASCII output file:

- Binary head file (modelname.HDS)
- Binary drawdown file (modelname.DDN)
- Binary cell-by-cell flow/budget file (modelname.CBC)
- ASCII listing file (modelname.LST)

The binary files contain head, drawdown, and flow exchange values for each grid cell, while the ASCII listing file contains all relevant information on the operation of MODFLOW-USG, and the simulation results. The listing (.LST) file is useful if errors occur during a simulation and you want to know how far MODFLOW-USG progressed, or if you want to examine head or drawdown values at given intervals.

For a steady-state simulation, only one set of values for each grid cell are written to these files. However, for transient simulations, each grid cell may contain simulation results for every time step within a given stress period, resulting in files that can become unnecessarily large. By default, the information is saved in the binary files at the end of each stress period and at the end of the simulation in the listing (.LST) file. Note that the cell-by-cell flows are saved by default for each time step if ZoneBudgetUSG or MOD-PATH3DU are included in the simulation.
The first two columns list the available stress periods and associated time steps for the entire simulation (only one stress period and time step will be listed for steady-state simulations). The remaining columns indicate the information which can be written and saved to the various MODFLOW output files. To select an output option, click in the appropriate checkbox and a checkmark (✓) will appear to indicate that the selected information will be written for the selected time step.

The columns labeled Save to Binary will save the output information to the binary files as described below.

- **Heads**: Saves the heads in the binary heads file (.HDS).
- **DDown**: Saves the drawdown in the binary drawdown file (.DDN).
- **F.Term**: Saves the cell-by-cell flow terms in the binary budget (.CBC) file.

⚠️ **Please Note**: The Zone Budget program uses the .CBC file for calculating the flow between zones. Therefore, to change the frequency at which the Zone Budget information is saved, select the desired F.Term intervals saved to binary.

The columns labeled Print to .LST will save the output information to the listing file as described below.

- **Heads**: Saves the heads in the listing file.
- **DDown**: Saves the drawdown in the listing file.
- **F.Term**: Saves the flux terms (cell by cell flow terms) in the listing file.
- **Budget**: Saves the budget information in the listing file.

⚠️ **Please Note**: MODFLOW only allows the flow terms (F.Term) to be stored once, in either the binary budget file (.CBC) or the listing file (.LST). Be aware that this setting can be lost if MODFLOW-USG is being run together with MOD-PATH3DU, because MOD-PATH3DU requires the flow terms to be written to the .CBC file, and not to the .LST file.

### Saving Output Every Nth Time Step

For simulations with many stress periods and time steps, it can be very tedious to manually select the desired output time step intervals. The row of fields underneath the Output Control table are used to specify regular time step intervals for saving files during Each N-th step in each stress period. The first text box is where the N value is entered. To apply this value to the column, click the underlying checkbox.

If a subsequent engine is included in the run formation (e.g. ZoneBudgetUSG, MOD-PATH3DU) with the MODFLOW-USG simulation, Visual MODFLOW Flex will save the flow terms for all time steps by default.
Advanced Settings

Advanced translation settings are available for flow simulations to enable more advanced control of translation and package settings. General and package-specific advanced settings are described below. General advanced settings are available for all Flow and Transport engines supported by Visual MODFLOW Flex, while certain engines have additional package-specific advanced settings available:

- **General**
- **MODFLOW-USG**

### Advanced Settings - General

The following settings are available for all packages:

- **CUnit**: The index for the package in the CUNIT array of the BAS file [read-only]
- **Ext**: The file extension of the package [read-only]
- **LUnit**: The file unit number to which the package will be read/written during the engine run [read-only]
- **Run**: Yes/No switch to determine whether the package will be added to the .NAM/.IN file and run during the simulation. **Note**: certain packages are required as part of a simulation (e.g. BAS file) and must always be run - the run option for these packages is disabled.
- **Translate**: Yes/No switch to determine whether the package will be translated. **Note**: packages that do not have sufficient data will not be translated (e.g. a river package file will not be translated if no rivers are listed in the model explorer in the active numerical model workflow).
Please Note: If a supported package is not represented in the model, by default it will not be translated and run, even if translate and run are both set to "Yes". For example if there are no drains represented in the model, then the DRN package will not be translated and run.

Working with Custom/Externally Developed Input Package Files
If you develop or modify a package input file outside of Visual MODFLOW Flex, you can still use it as part of your modeling workflow within the Flex environment. This is particularly useful if you want to use one or more packages that are not yet supported in Visual MODFLOW Flow or if you have custom methods/utilities to generate input files. To use a custom input file:

1. Set Run = Yes
2. Set Translate = No
3. Add the input file to the model run folder with the correct name and extension: [model].[ext]. The model run folder can be found by selecting the folder icon button [], the model name will correspond to the name of your conceptual model, and the extension will be the Ext value for the package in question. For example, if you will to run the drain return package (DRT), and you are running a model where:
   - the Visual MODFLOW Flex project file is called SiteModel.amd,
   - the Conceptual Model is called Concept1,
   - the Grid is called NumericalGrid,
   - the Run is called Run 1, and
   - the Flow Engine is MODFLOW-2005

Then the DRT file must be saved as:

.\SiteModel.data\MODFLOW\NumericalGrid\Run1\MODFLOW-2005\Concept1.DRT

If the above conditions are met, the custom package file will inserted into the relevant .NAM/.IN file and run with the model. Note that certain results from unsupported packages will not be read back into Visual MODFLOW Flex, such as the flow balance terms associated with the SFR package.

Advanced Settings - MODFLOW-USG
Package-specific settings for MODFLOW-USG are described below:

OC Settings
The following settings are specific to the Output Control (OC) package:
Output control file format type - An integer value used to define which cell in a vertical column that recharge and discharge is simulated.
- use codes - the output control file will be generated with integer codes to that mark what output will be generated for each time step
- use keywords [Default] - the output control file will be generated with keywords that are more easily legible. This option also enables the use of the compact budget format.

Use compact budget - an optional keyword that controls the format of the output binary file
- Yes [Default] - the compact budget format will be used. This format uses less disk space than the standard format developed with MODFLOW-88
- No - the standard format developed with MODFLOW-88 will be used.

CHD Settings
The following settings are specific to the specified head (CHD) package:

Duplicate CHD Cell Filter - In some cases multiple constant head cells may be assigned to the same cell, particularly when working with conceptual models. When MODFLOW formulates the system of equations to be solved, it simply sequentially adds boundary condition terms to the matrix and/or solution vector - for all other boundary conditions this is usually not a problem as these values still require a head solution for that cell; however adding the constant head values together typically does not produce the intended (i.e. without intervention, overlapping constant head boundary conditions in the same cell with values of 201 and 199 would result in an output head value at that cell of 400, which is most likely an undesired result). Therefore, Visual MODFLOW Flex includes an optional filter to handle these cases on translation:
- Minimum - the minimum specified head value for the current stress period in the cell(s) with multiple values will be assigned
- Maximum - the maximum specified head value for the current stress period in the cell(s) with multiple values will be assigned
- Average [Default] - the arithmetic mean of specified head values for the current stress period in the cell(s) with multiple values will be assigned
- Sum - specified head values values for the current stress period in the cell(s) with multiple values will be added.
### GNC Settings

The following settings are specific to the ghost-node correction package:

<table>
<thead>
<tr>
<th>GNC</th>
<th>0 - Do not apply second-order correction</th>
<th>1 - Apply second order correction</th>
</tr>
</thead>
<tbody>
<tr>
<td>I2KN</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ISYMGNCn</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **I2KN** - a flag indicating if second-order correction is also applied to the unconfined transmissivity term by using the smoothed function for expressing the downstream node head for “flow-to-dry-cell” option ($\bar{h}$) to compute the transmissivity of the ghost node instead of using $h$.
  - 0 - Do not apply second order correction [Default]
  - 1 - Apply second order correction

- **ISYMGNCn** - a flag indicating if the GNC formulation is applied in an implicit manner on the left-hand side matrix for an asymmetric system, or in an explicit manner on the right-hand side vector with iterative updates as is required for symmetric systems.
  - 0 - Use implicit update for asymmetric formulation [Default]
  - 1 - Use explicit update for asymmetric formulation

### 11.7.2 MOD-PATH3DU Translation Settings

The mod-PATH3DU program is used for calculating the advective flow pathlines and travel times for forward tracking and backward tracking particles for both finite difference and unstructured grids.
mod-PATH3DU has run-time settings for:

- **IFACE**: settings to assign boundary condition fluxes to cell faces:
  - [0] Internal sink/source;
  - [2] Implicit side-face; flow is not explicitly assigned to a cell face, rather mod-PATH3DU distributes the flow to any side faces with zero flow;
  - [5] Same as MODPATH. Flow through the bottom face of a cell;
  - [6] Same as MODPATH. Flow through the top face of a cell;
  - [7] Internal sink/source. Flow is assigned to the top face.

- **ThreadCount**: The number of Threads to use for processing particle tracks
- **Mp3duSimulations**: Opens a dialog for assigning settings for individual particle groups:
Each particle group (shown on the left side of the dialog) has the following set of parameters (shown on the right) specified:

- **Adaptive Step Error**: The error criterion used to determine the maximum allowed error in all directions for the adaptive step size procedure.
- **Capture Radius**: The radial distance from within which a particle will be considered captured by a pumping well.
- **Direction**: This parameter is set when creating/editing the particle group and is disabled.
- **Euler DT**: The step size below which the Euler method is used instead of the Runge-Kutta method.
- **Initial Step Size**: The initial particle tracking stepsize
- **Max Step Size**: The maximum particle tracking stepsize
- **Name**: The name of the particle group
- **Simulation End Time**: Can be used to overwrite the simulation end time in the model. The default value is the flow model end time for forward simulations and 0 for backward simulation.
- **Stagnation DT**: The tracking size below which a particle is considered stagnated.
- **Run**: Whether or not Flex will translate and run the particle group
- **Track To Termination**: Track the particles in the group on the last time step until each particle terminates at a boundary or is removed by reaching a stagnation point.
Tip!

Editing Settings for Multiple Particle Groups

You can select one or more particle groups using the mouse and <SHIFT> or <CONTROL> keys and then adjust these parameters in for the selected groups in bulk.

Please refer to the mod-PATH3DU documentation for more information about the settings listed above.

11.8 Run Engines

At this step, run the MODFLOW-USG engine. During a run you will see the Stress Period and Time Step count. Upon a successful run, you will see a budget summary from the .LST file:

```
<table>
<thead>
<tr>
<th>Cumulative Volumes</th>
<th>L^3/y</th>
<th>Rate for this Time Step</th>
<th>L^3/y</th>
</tr>
</thead>
<tbody>
<tr>
<td>IN:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>STORAGE</td>
<td>0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CONSTANT head</td>
<td>1220.3724</td>
<td></td>
<td>1220.3724</td>
</tr>
<tr>
<td>RIVER LEAKAGE</td>
<td>9125589.0000</td>
<td></td>
<td>9125589.0000</td>
</tr>
<tr>
<td>TOTAL IN</td>
<td>9126794.2724</td>
<td></td>
<td>9126794.2724</td>
</tr>
<tr>
<td>OUT:</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>STORAGE</td>
<td>0.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CONSTANT head</td>
<td>3.1678623</td>
<td></td>
<td>3.1678623</td>
</tr>
<tr>
<td>RIVER LEAKAGE</td>
<td>9123590.0000</td>
<td></td>
<td>9123590.0000</td>
</tr>
<tr>
<td>TOTAL OUT</td>
<td>9123600.0002</td>
<td></td>
<td>9123600.0002</td>
</tr>
<tr>
<td>IN - OUT</td>
<td>-0.7308</td>
<td></td>
<td>-0.7308</td>
</tr>
<tr>
<td>Percent Discrepancy</td>
<td>0.00</td>
<td></td>
<td>0.00</td>
</tr>
</tbody>
</table>
```

11.9 View Results

At the View Results step, you have the option to View Maps or View Charts.

At the View Maps step you can visualize output from the following model results:

- Heads, Drawdown, Velocity and
- Pathlines (if particle tracking was simulated).

View Charts contains several chart types:
- Observed vs. Calculated Heads
- Time Series
- Zone Budget
- Mass Balance.

11.9.1 View Maps (USG)

The options for interpreting results from a MODFLOW-USG run are very similar to the options from a finite difference (e.g. MODFLOW-2005) run. The section below describes the options and settings that are specific to working with unstructured grids:

Heads and Drawdown
Heads and Drawdown display settings are identical to those that you have for Property zone objects, as described in USG: View and Edit Properties.

Velocity
At the View Maps step, you can view groundwater velocities in a variety of ways as described in the section on 3D Gridded Data Settings. By default, velocity vectors are shown as vectors.
**Water Table**

Visual MODFLOW Flex also calculates the water table elevation following a MODFLOW-USG run; simply turn on this data object from the Model Explorer under the Output node, and this will appear as a Surface in the 3D viewer. The Water Table has style settings similar to a surface for controlling the color classes and contour lines.

**Export**

Heads and Drawdown can also be exported to point or polygon shapefile, along with the attributes. The options are similar to what is described for Structured MODFLOW models; see Exporting Data for more details.

The Water Table data can also be exported to .CSV file for further post-processing. Right-click on Water Table from the Model Explorer and select "Export..."

The format of this file is as follows:

- x (x-coordinate at center of cell)
- y (y-coordinate at center of cell)
- ztop (elevation of the top of layer 1; this can be used to calculate depth to water table)
- nodeId: MODFLOW-USG global node number
- wt_"time": water table elevation at the defined time step; for transient models, you will see additional columns for each saved MODFLOW time step.
12 Expression Builder

The field calculator is available in the Edit property dialog to facilitate creating/editing variable or distributed values of properties within each zone. Mathematical expressions including variable names can be typed directly into the ACalculation entry form (show in red below) or with the assistance of the Expression Builder ( button).

Building Expressions

The Expression Builder can be used to construct mathematical expressions to specify variable property (e.g. Kx, initial heads, Kd) and boundary condition (e.g. river conductance, constant head, etc.) values. The Expression Builder is shown below for editing conductivity values. The Expression builder consists of five components:

1. **Expression workspace**: a workspace to build expressions;
2. **Operators**: basic arithmetic operators (e.g. addition, subtraction, multiplication, division, parentheses, and exponents);
3. **Variables & Constants**: a list of variables associated with the current model cell and constants
4. **Data Objects**: a list of supported data objects from the data explorer and model explorer windows
5. **Functions**: a list of supported mathematical, logical, and vector functions
Please Note: The expression builder is case sensitive.

Operators

The following operators are always available in the Expression Builder:

<table>
<thead>
<tr>
<th>Operator</th>
<th>Button</th>
<th>Example</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Add</td>
<td>+</td>
<td>1+2</td>
<td>3</td>
</tr>
<tr>
<td>Subtract</td>
<td>-</td>
<td>4-3</td>
<td>1</td>
</tr>
<tr>
<td>Divide</td>
<td>÷</td>
<td>2/4</td>
<td>0.5</td>
</tr>
<tr>
<td>Multiply</td>
<td>×</td>
<td>2*5</td>
<td>10</td>
</tr>
<tr>
<td>Open bracket</td>
<td>(</td>
<td>3*(5+2)</td>
<td>21</td>
</tr>
<tr>
<td>Close bracket</td>
<td>)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Power</td>
<td>x⁵</td>
<td>5³</td>
<td>125</td>
</tr>
</tbody>
</table>
Please Note: The expression builder does not support spaces between operators.

Variables & Constants
The expression builder includes a list of applicable model properties or boundary condition values as well as standard constants as described below.

Grid Geometry Variables
The following grid-based variables are always available in the Expression Builder:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$BOT</td>
<td>cell bottom elevation</td>
</tr>
<tr>
<td>$DX^1</td>
<td>cell width in the x-direction</td>
</tr>
<tr>
<td>$DY^1</td>
<td>cell length in the y-direction</td>
</tr>
<tr>
<td>$DZ</td>
<td>cell thickness</td>
</tr>
</tbody>
</table>

Notes:
1 - finite difference grids only

Editing Model Properties
The following variables are available in the Expression Builder during Model Property editing sessions:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$OLD{fieldname}</td>
<td>initial value of the specified field {fieldname} at the start of the edit session (e.g. $OLDINITIALHEADS for the initial heads property field)</td>
</tr>
</tbody>
</table>

Assigning Boundary Conditions By Geometry
The following variables are available in the Expression Builder when assigning Boundary Conditions using polylines or polygons or using a (polyline or polygon) data object:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$FACEAREA_NORTH</td>
<td>area of the current cell projected onto the model-coordinate XZ plane</td>
</tr>
<tr>
<td>$FACEAREA_TOP</td>
<td>area of the current cell projected onto the XY plane (i.e. plan area)</td>
</tr>
<tr>
<td>$FACEAREA_WEST</td>
<td>area of the current cell projected onto the model-coordinate YZ plane</td>
</tr>
</tbody>
</table>
### Variable | Description
--- | ---
$RCHLNG$ | length of the intersection between the current cell and the input polyline
$LENGTH$ | percentage along the length of the polyline from the beginning of the polyline to the exit point of the polyline from the current cell
$SEGMENT$ | ID of the polyline intersecting the cell
$UCTOCOND$ | unit conversion factor for converting the length and time units used for hydraulic conductivity to the length and time units used for conductance

**Notes:**

$^1$ - polyline features only

---

### Editing Boundary Conditions

The following variables are available in the Expression Builder during Boundary Condition editing sessions:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$KX$</td>
<td>horizontal conductivity along X-axis</td>
</tr>
<tr>
<td>$KY$</td>
<td>horizontal conductivity along Y-axis</td>
</tr>
<tr>
<td>$KZ$</td>
<td>vertical conductivity</td>
</tr>
<tr>
<td>$MBOT$</td>
<td>model bottom elevation</td>
</tr>
<tr>
<td>$OLD{fieldname}$</td>
<td>initial value of the specified field {fieldname} at the start of the edit session (e.g. $OLDCOND$ for river conductance values)</td>
</tr>
</tbody>
</table>

---

### Constants

The following constants are always available in the Expression Builder:

<table>
<thead>
<tr>
<th>Keyword</th>
<th>Constant</th>
<th>Example</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>false</td>
<td>False (constant)</td>
<td>false</td>
<td>0</td>
</tr>
<tr>
<td>pi</td>
<td>Pi (π) (constant)</td>
<td>pi</td>
<td>3.141592653</td>
</tr>
<tr>
<td>true</td>
<td>True (constant)</td>
<td>true</td>
<td>1</td>
</tr>
</tbody>
</table>

---

© 2019 by Waterloo Hydrogeologic
Data Objects

The Data Objects component of the Expression Builder includes a list of objects from the Data Explorer and the Model Explorer windows that can be included in expressions. Surfaces and horizons are currently supported. Values from the surface/horizon are bilinearly interpolated to the location of the cell centroid (in the case of finite difference grid and quadtree grids) and to the interior cell node (in the case of Voronoi grids). Note that bilinear interpolation is extrapolated beyond the extents of a given surface.

The syntax to use a surface is $`object_name` where `object_name` is the name of the surface in the Data Explorer or the name of the horizon in the Model Explorer, as applicable. Surfaces may be used in expressions:

Examples

- The expression $`Ground`-1 would result in a value of 1 unit of length below ground surface. This may be useful when defining features such as the bottom elevation of a river.

- The expression pmin($`Ground` $`Pit Elev`) would result in the lower elevation between ground surface and a surface representing the elevation of an excavated pit.

Functions

The following functions are always available in the Expression Builder:

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
<th>Example</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>abs</td>
<td>Absolute Value</td>
<td>abs(-1.2)</td>
<td>1.2</td>
</tr>
<tr>
<td>average</td>
<td>Average</td>
<td>average({1 2 3})</td>
<td>2</td>
</tr>
<tr>
<td>cif</td>
<td>Conditional If</td>
<td>cif(true 23 24)</td>
<td>23 24</td>
</tr>
<tr>
<td></td>
<td></td>
<td>cif(false 23 24)</td>
<td>23 24</td>
</tr>
<tr>
<td></td>
<td></td>
<td>cif(1 23 24)</td>
<td>23 24</td>
</tr>
<tr>
<td></td>
<td></td>
<td>cif(0 23 24)</td>
<td>23 24</td>
</tr>
<tr>
<td></td>
<td></td>
<td>cif(1-1 23 24)</td>
<td>24</td>
</tr>
<tr>
<td>cos</td>
<td>Cosine</td>
<td>cos(0)</td>
<td>1</td>
</tr>
<tr>
<td>dot</td>
<td>Dot Product</td>
<td>dot({1 2 3} {1 2 3})</td>
<td>14</td>
</tr>
<tr>
<td>equal</td>
<td>Equality</td>
<td>equal(2+2 4)</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
<td>equal(2+2 5)</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>cif(equal(2+2 5) 23 24)</td>
<td>24</td>
</tr>
<tr>
<td>Function</td>
<td>Description</td>
<td>Example</td>
<td>Result</td>
</tr>
<tr>
<td>----------</td>
<td>-------------</td>
<td>---------</td>
<td>--------</td>
</tr>
<tr>
<td>exp</td>
<td>Exponential</td>
<td>exp(1)</td>
<td>2.71828182845905</td>
</tr>
<tr>
<td>ln</td>
<td>Natural logarithm</td>
<td>ln(2.718218)</td>
<td>1</td>
</tr>
<tr>
<td>log</td>
<td>Logarithm</td>
<td>log(16 4)</td>
<td>2</td>
</tr>
<tr>
<td>log10</td>
<td>Base-10 logarithm</td>
<td>log10(100)</td>
<td>2</td>
</tr>
<tr>
<td>max</td>
<td>Maximum Value</td>
<td>max({10 2.2 34})</td>
<td>34</td>
</tr>
<tr>
<td>min</td>
<td>Minimum Value</td>
<td>min({10 2.2 34})</td>
<td>2.2</td>
</tr>
<tr>
<td>mod</td>
<td>Modulo</td>
<td>(10)mod(3)</td>
<td>1</td>
</tr>
<tr>
<td>mult</td>
<td>Multiplication</td>
<td>mult({2.0 3.0 4.0})</td>
<td>24</td>
</tr>
<tr>
<td>not</td>
<td>Negation</td>
<td>not(true) not(false) not(1) not(0) not(2) not(1-1)</td>
<td>0 1 0 1 0 1</td>
</tr>
<tr>
<td>pmax</td>
<td>Maximum Value (pairwise)</td>
<td>pmax(1 2)</td>
<td>2</td>
</tr>
<tr>
<td>pmin</td>
<td>Minimum Value (pairwise)</td>
<td>pmin(1 2)</td>
<td>1</td>
</tr>
<tr>
<td>round</td>
<td>Round to Decimal Place</td>
<td>round(1.523 2) round(5.523 0)</td>
<td>1.52 10</td>
</tr>
<tr>
<td>sigdig</td>
<td>Round to significant digit</td>
<td>sigdig(1523 2) sigdig(1.523E+3 1)</td>
<td>1500 2000</td>
</tr>
<tr>
<td>sin</td>
<td>Sine</td>
<td>sin(pi/2)</td>
<td>1</td>
</tr>
<tr>
<td>sqrt</td>
<td>Square Root</td>
<td>sqrt(4)</td>
<td>2</td>
</tr>
<tr>
<td>sum</td>
<td>Sum</td>
<td>sum({1 2 3})</td>
<td>6</td>
</tr>
<tr>
<td>tan</td>
<td>Tangent</td>
<td>tan(pi/4)</td>
<td>1</td>
</tr>
<tr>
<td>vector</td>
<td>Vector</td>
<td>vector({3 2})</td>
<td>{2.0 2.0 2.0}</td>
</tr>
<tr>
<td>Function</td>
<td>Description</td>
<td>Example</td>
<td>Result</td>
</tr>
<tr>
<td>----------</td>
<td>-------------</td>
<td>---------</td>
<td>--------</td>
</tr>
</tbody>
</table>
| vequal   | Vector Equality | vequal({1 2 3} 
{1 2 3})

cif(vequal({4 5 6} 
{4 6 6}) 10 20) | true 20 |

Examples are also provided in the help text, which you can view by hovering the mouse pointer over a given function.
13 **PEST (Parameter Estimation) Workflow**

Visual MODFLOW Flex provides a seamless interface to the popular parameter estimation and predictive analysis program PEST, developed by Dr. John Doherty of Watermark Computing. This section provides instructions on using Visual MODFLOW Flex to setup, run, and interpret a Parameter Estimation/Predictive Analysis simulation. In addition, this chapter provides a brief description of the input parameters and settings required by PEST. A detailed description of the algorithms, parameters, input files, and other options for PEST are available in the PEST User Documentation. This can be accessed from [www.PESTHomepage.org](http://www.PESTHomepage.org)

---

### Before you start

You are encouraged to familiarize yourself with the concepts and applications of PEST prior to using in Visual MODFLOW Flex. The time spent on this will make your experience with parameter estimation much more productive, and will likely help you to overcome any difficulties you may experience the first time you run PEST.

### Note

You must have a license of Pro or Premium in order to use the PEST module in Visual MODFLOW Flex.

Visual MODFLOW supports both the Calibration and Predictive Analysis capabilities of the PEST program, and it allows you to run parameter estimation using results from both groundwater flow and contaminant transport simulations (i.e. observations can consist of heads, concentrations, and groundwater flux). Support for transport, will be added in future releases.

---

**Acknowledgements:**

Excerpts from the following publications are used throughout this documentation:

- Using Pilot Points to Calibrate a MODFLOW/MT3D Model, Doherty *(2008)*
- Addendum to the PEST Manual, June 2012
- [www.PESTHomepage.org](http://www.PESTHomepage.org)
Suggested References:

Several USGS publications are also available on PEST. See:

Approaches to Highly Parameterized Inversion: a Guide to Using PEST for Groundwater Model Calibration:

Approaches to Highly Parameterized Inversion: a Guide to Using PEST for Model Parameter and Predictive Uncertainty Analysis

Approaches to Highly Parameterized Inversion: Pilot Point Theory, Guidelines and Research Directions

Loading the PEST Workflow

The PEST workflow can be launched from the “Select Run Type” step in a numerical workflow, as shown below; simply click on the PEST button in the main window.

Before attempting to run a parameter estimation simulation, make sure your model meets the following requirements:

- The model runs successfully (converges) and produces meaningful results. The model should provide a solution under a variety of input parameter conditions. Parameter estimation is as much an art as it is a science, and therefore, it should only be used to complement your own efforts in understanding the system.
The model has one or more (preferably many more) observations against which to compare the calculated results. Observations can be in the form of measured or estimated values of head or concentration at discrete points in the model, or in the form of measured or estimated groundwater fluxes into (or out of) one or more grid cells.

It is also recommended that you do a sensitivity analysis prior to a full-fledged PEST run. In a sensitivity analysis, parameter values are individually changed to determine the effect on model calibration and prediction. The results give an indication of which parameter changes can have significant impact on the model results (these are sensitive parameters) and which parameter changes have little or no impact on the model results (these are non-sensitive or insensitive parameters). This should help you to properly select parameters to include in a PEST run, as you should focus more on the sensitive parameters, and less on the non-sensitive parameters.

The observation times (for heads) lie within the start/stop time of the numerical simulation. If you have imported a model from Visual MODFLOW Classic, ensure that an appropriate start date was defined in Visual MODFLOW Classic before loading the project into Visual MODFLOW Flex. If you have generated the numerical model from a conceptual model workflow, ensure that the start date at the modeling objectives is appropriate based on the head observation times you have defined.

Setting up a PEST Run

PEST requires several inputs and a number of steps that must be completed in a specific order. Fortunately, Visual MODFLOW Flex provides the PEST GUI in a workflow that guides you through the sequential steps and necessary inputs, running PEST, and analyzing the results:

- **Define Observations** and Assign Weights
- **Define Parameters** (Property Zones, Boundary Conditions)
- **Define Pilot Points**
- **Define Kriging Variograms**
- **Select Regularization** (None, Tikhonov, SVD Assist)
- **Define PEST Run Settings** and Run PEST
- **Run Sensitivity Analysis**
- **Analyze Results**
- **Update Model Inputs**

13.1 Define Observations

When PEST is used to assist in the calibration of a model, it is tasked with minimizing an objective function, represented by the symbol $\Phi$. This is comprised of the sum of the weighted squared deviations between the calculated ($X_{\text{calc}}$) and observed ($X_{\text{obs}}$) system responses and is ultimately a function of the defined Parameters ($p_i$). The difference between the calculated and observed values are referred to as a residual, and represented by the following equation:
In the simplest PEST run, the objective function ($\Phi$) is calculated using the following equation:

$$\Phi = \sum_i^n\min(r_i) = f\left(p_j\right)$$

In the case of a groundwater model, these system responses are typically head at a point in space, concentration at a point in space, or groundwater flow to a specified zone.

At the Define Observations step, you will see a list of all your available head observations as shown in the following figure:

The observation values are assigned a default weight of 1.0, which means the residual will not be modified, and all observations will be treated equally.

We want PEST to:

- give more weight to measurements that you can trust
- give greater priority to measurements in areas where predictions are important
- decrease bias that is introduced by denser data (if appropriate)
• give equal weight to observations with different units (eg. Heads vs. Fluxes, vs Concentrations)

The weight factor is commonly used to normalize observations of different types. For example, when you have head and flux observations included in the same PEST run, the flux residuals are typically much higher than a head residual. For example:

- a flux residual for a set of cells could be +/- 1000 m3/day
- a head residual at a single cell is +/- 1m.

PEST calculates the objective function by summing the squares of these residuals, which results in flux observation residuals being several orders of magnitude greater than the head residuals. As a result, PEST will focus more on minimizing the flux residuals, and less so on the head residuals, which is most likely undesired from the modeler’s view point.

The solution is a user-defined weighting factor \(w_i\)

\[
\Phi = \sum_{i}^{n} \min(w_i r_i)
\]

We use a weight less than 1.0 to reduce the magnitude of flux errors so they are comparable to head errors. A similar problem is often seen with concentration targets, for the same reason. The ability to use weights to address a variety of common calibration issues makes the objective more useful.

**Settings**

If you click on the [Setting] button above the Observations grid, you can adjust the Time Extrapolation Limit; this value is used by PEST to search for the nearest observation time for comparing to calculated MODFLOW output times. The higher the value, the more likelihood that an observation time can be found to match the MODFLOW output times. The default is 100 (in the project time units).

**Please Note**: the value of the Time Extrapolation Limit is particularly important for matching observations with Steady State models as the time for comparison is the steady state simulation period. If you have a particularly short (e.g. 1 day) or long (e.g. 10 year) steady state simulation period, your PEST analysis may not have any valid observations (or fewer than expected), depending on the model start date relative to the date(s) of the observations.

**13.2 Define Parameters**

At the Define PEST Parameters step, select which model input types you wish to include in the PEST run.
Parameter Settings

In the table at the top of the window, select which parameter(s) you want to include; Visual MODFLOW Flex currently supports the following parameters:

- Conductivity (Kx and Kz)
- Storage (Ss and Sy)

Please Note: Ss and Sy should only be included in a transient MODFLOW run

This list of available parameters will be extended in future versions to include more parameters from both the flow and transport models.

Select the check-box in the Use column in order to include this parameter zone.

Tied To

- Tied: The parameter value is tied (linked) to the value of another parameter, in which case this parameter takes only a limited role in the parameter estimation process. If you select the tied to option, then you can select the parent parameter from the dropdown list.

Please Note: you can only tie to other parameters that have been selected/included in the PEST run). PEST does not allow a parameter to be tied to a fixed parameter, or a parameter that is already tied to another parameter.
If a parameter is tied to a parent parameter, the parameter “piggy-backs” on the parent parameter during the estimation process. In this way, the ratio between the initial values of the parameter and its parent remain constant throughout the estimation process.

**Transformation**

In many cases, the linearity assumption on which PEST is based is more valid when certain parameters are log-transformed. This means that the log-transformation of some parameters can often make the difference between success and failure of the estimation process. However, a parameter that can become zero or negative during the estimation process must not be log-transformed.

The parameter Transformation field controls how the parameter value will be transformed during the optimization process. There are three transformation options:

- **Fixed**: The parameter value is fixed and takes no part in the parameter estimation process.
- **Log**: The parameter will be log-transformed (i.e., the log value of the parameter is optimized rather than the ‘real’ value of the parameter).
- **None**: No transformation takes place (i.e., the ‘real’ parameter value is adjusted)

If the parameter is log-transformed then the covariance, correlation coefficients and eigenvector values refer to the log of the parameter. However, the parameter estimates and confidence intervals refer to the untransformed parameter.

If you fix a parameter, its value will be fixed at its initial value, and it will not be part of the estimation process.

**Parameter Zone Settings**

In the table at the bottom of the window, select which parameter zones you wish to include; by default, all zones from a selected parameter type will be included. The Value column indicates the value for each zone; in the case where you have distributed parameter values in a zone, you will see “Distributed” in this column. The Minimum and Maximum are the lower and upper bounds of the parameter zone.

The lower and upper bounds should be chosen wisely; for example, if you know that a zone represents an sandy-gravel aquifer, then define practical limits for the upper and lower bounds for conductivity for sandy-gravel. The default values are 1E-15 and 1E+30 respectively. The lower and upper bounds are ignored for fixed and tied parameters. If an updated parameter value is outside of its bounds, PEST temporarily holds the parameter at its boundary value.
Updating Parameter Values after a PEST Run

If you have completed a PEST run and return to the Define Parameters step to make a change (such as add/remove zones or adjust the Minimum or Maximum values), you need to click the [Apply] button on the workflow toolbar, and also proceed through the Define Pilot Points step, and re-generate the Define Kriging Variograms step in order to update the appropriate input files.

13.3 Define Pilot Points

The next step is to define Pilot Points.

Pilot points are simple XY points with an initial value for each parameter that you want PEST to estimate. Pilot points can be imported from .TXT file, XLS, .SHP file, or assigned manually by digitizing in the 2D environment. An example of the pilot points on top of conductivity zones is shown below.
The general steps are as follows:

- Select the desired points object from the model tree
- Click (Next Step button) to add these points.
- Select which parameter zones the points represents (eg. Kx-Zone1, or Kz-Zone5)

In the table in the lower half of the window, specify which pilot points are "Fixed". Fixed pilot points are those that have initial values that are well known, and will not be adjusted during the PEST run. Examples of these include points where you have pumping/slug test data. For all other points, their initial value is taken from the property zone in which the specific point lies.

Visual MODFLOW Flex handles these "Fixed" pilot points the following way: In the PEST Control file,

- these pilot points parameters use "Fixed" Transformation, and the initial value defined in the table above will appear beside each point; and
- the regularization information for these fixed parameters is not included under the *prior information section. *This is only applicable when one of the regularization options are used in the PEST run.*

Once you are finished, be sure to repeat these steps for the other parameter zones you have included.
Theory

Using conventional model calibration technology, the calibration process would use one parameter associated with a property zone, and adjusting these parameters until the fit between model outcomes and field observations is as good as possible. If the goodness of fit obtained on the basis of these zones was not acceptable, then extra zones would be introduced into the model domain at locations where the modeler felt that they would “do the most good”; the parameterization process would then be repeated with the new parameters included. If a good fit was still not obtained, more zones would then be introduced. The process would continue until the fit between model outcomes and field observations was acceptable.

There are a number of shortcomings associated with this approach. These include the following:

- The procedure is quite laborious and slow.
- In a case such as the present one, geological mapping provides no guidance on where to put extra zones; hence the final distribution of zones that the modeler achieves using this process is likely to be subjective and non-unique.
- Characterization of geological heterogeneity in the present study area by zones of piecewise uniformity is not in harmony with the nature of the alluvial material; therefore any zonation pattern that is finally decided upon will not “look right”; it will be defensible only on the basis that it is better to employ such a zonation scheme than to ignore geological heterogeneity altogether.
- Piecewise constancy as a method of characterizing geological heterogeneity lacks the flexibility required to explore the effects of small scale geological variability on model predictive uncertainty.

To overcome these problems, the distribution of transmissivity within the model domain will be described by a set of pilot points. A number of these pilot points will be introduced to the model domain and PEST will be asked to estimate the transmissivity of the aquifer at each such point. These “point-transmissivities” will then be spatially interpolated to all of the active cells within the model domain using Kriging. Hence in estimating transmissivity values at pilot points, PEST will effectively be assigning parameter values to the whole of the model domain.

Individual pilot points can be assigned to different zones within the model domain. Only those points assigned to a particular zone can be used in calculating transmissivity values throughout that zone using the Kriging interpolation procedure. Furthermore, the variogram upon which Kriging is based can be different in each zone, reflecting differences in the geology, or in the level of heterogeneity, expected within each geological unit. Note that if only
one pilot point is assigned to a particular zone, then that zone is characterised as being uniform.

Conventional wisdom dictates that the number of parameters involved in a parameter estimation exercise should be kept to a minimum. However when using pilot points in conjunction with PEST’s new regularization mode, the opposite is often true. PEST’s regularization functionality prevents the onset of numerical instability which often accompanies attempts to solve over-parameterised inverse problems. Furthermore, a superfluity of pilot points allows PEST to achieve a much better fit between model outcomes and field observations.


General Tips

General tips on using pilots points (excerpted from http://www.pesthomepage.org/PEST_FAQ.php)

- There is no need to be parsimonious; use as many pilot points as you can. Let PEST do the regularization (use SVD, Tikhonov, SVD-assist or all of these together).

- Use more pilot points in those parts of the model domain where information has a high spatial density (for example where there are more observation wells).

- Try to place pilot points between head measurement wells in the direction of groundwater flow where they represent hydraulic conductivity. Place them on measurement wells where they represent specific yield or storage coefficient.

- Make sure you place hydraulic conductivity pilot points between measurement wells and any downstream boundary, for it is the hydraulic conductivity of the material between these wells and the boundary that determines the heads in those wells.

- Don’t leave any large gaps in the final pilot points pattern.

- If you use preferred value regularization (i.e. you have selected Preferred Value Tikhonov Regularization) consider using a covariance matrix instead of weights for the prior information equations. See the PPCOV utility from the Groundwater Data Utilities suite.

- If you are absolutely sure that no heterogeneity can arise over a distance of less than x, then don’t place any pilot points closer together than a distance of x.

13.4 Define Kriging Variograms

Once the pilot points are generated, the next step is to specify the Kriging settings that will be used to interpolate the pilot point data during PEST calibrations for these zones. Principally,
this involves definition of the appropriate variogram. Variograms are also used by PEST in the calculation of regularization weights and so are an important part of the process.

Visual MODFLOW Flex will create one default variogram for each parameter zone "structure".

Visual MODFLOW Flex uses Ordinary Kriging to interpolate between pilot points. This method has the following advantages:

- numerically efficient
- respects values assigned to the points
- regularization and interpolation can be based on same variogram

The only downside is that it requires a defined variogram to be effective and each parameter+zone must have its own variogram.

In Visual MODFLOW Flex, a default variogram is provided for each parameter+zone. Suggested defaults are:

- Exponential Variogram Type
- A = approx 20% of the length or width ( whichever is greater) of the model domain; typical range is 20-70%
- Anisotropy = 1 (uniform)
- Nugget = 0
Select the desired Parameter zone from the tree, and provide the necessary variogram parameters.

**Please Note**: Gaussian variograms are not recommended for use with PEST because they can lead to spurious interpolated property fields greater than or less than the highest/lowest property value assigned to any pilot point (Doherty, 2008a).

**Theory**

**Spatial Interpolation using Pilot Points**

The use of pilot points in characterizing the spatial distribution of a hydraulic property must be accompanied by a mechanism whereby hydraulic property values assigned to pilot points are spatially interpolated to the cells of the finite difference grid. Spatial interpolation is accomplished using the Kriging algorithm. Kriging is a method of spatial interpolation based on geostatistics. The cornerstone of geostatistics is the variogram; a variogram describes the extent to which hydraulic property values (or any other type of data) pertaining to any two
points are likely to be different from each other as a function of the distance between those points.

One of the benefits of using Kriging as a basis for spatial interpolation is that the factors by which hydraulic properties at pilot points are multiplied before summation to obtain the hydraulic property value at a particular grid cell are independent of the actual hydraulic property values at the pilot points. Hence a set of “Kriging factors” pertaining to each of the cells of the finite difference grid can be calculated in advance of the actual interpolation process. As the latter is undertaken again and again as the model is run repeatedly by PEST, the fact that it is not necessary to repeat calculation of the Kriging factors on each occasion that the model is run can result in large savings in the time required to complete the overall parameter estimation process.

Calculation of Kriging factors is undertaken by program PPK2FAC. Variograms upon which these Kriging factors are based are supplied to PPK2FAC in a “structure file”. Such a file is struct.dat. Inspect this file using your screen editor. For full details of the specifications of this file, see the documentation to the Groundwater Data Utilities. Note that the variogram assigned to zones that have just a single pilot point is quite unimportant; because there is only one pilot point assigned to each of them, all cells within these zones will be assigned the one interpolated value (same as the respective pilot point) irrespective of the variogram.

Also note that for all of the structures appearing in file struct.dat, the TRANSFORM variable is set to “log”. Thus any variogram cited in each of these structures must pertain to the spatial distribution of the logarithm of the pertinent hydraulic property. This is in accord with the fact that most studies cited in the groundwater literature which treat transmissivity and/or hydraulic conductivity as a regionalized variable indicate that its distribution is better described by a log variogram than by a variogram based on native property values.

13.5 Select Run Type

At this step, you can select one of two types of PEST run:

- **Run Parameter Estimation** - for finding the optimal set of parameters, based on your observations

- **Run Sensitivity Analysis** - to calculate parameter sensitivities, which will help you to determine which parameters are influential or not-influential on the model results (based on the current values of the parameters).
13.5.1 Run Sensitivity Analysis

A sensitivity analysis is the first step in assessing the possible range of results that can be derived from the groundwater model.

In a sensitivity analysis, parameter values are individually changed to determine the effect on model calibration and prediction. The results give an indication of which parameter changes can have significant impact on the model results (these are sensitive parameters) and which parameter changes have little or no impact on the model results (these are non-sensitive or insensitive parameters). This should help you to properly select parameters to include in a PEST run, as you should focus more on the sensitive parameters, and less on the non-sensitive parameters.

When you go to Run Sensitivity Analysis, the following window will appear:

Under the "PEST Control file" tab, you can see the PEST control file has been automatically generated for you with the appropriate settings for a Sensitivity Run.
Click on the "Run PEST" button to start the PEST Run; you should see the PEST progress in a DOS window.

When the run completes, you will see several new tabs added to the workflow window.
Select the appropriate tab and you will see the sensitivities for the parameters and observations, along with other useful PEST run results.

13.5.2 Select Regularization

At this step, choose which regularization options you want to use in the PEST Run.

- No Regularization
- Tikhonov Regularization
- SVD Assist
Theory

One of the great advantages of using pilot points is that we can distribute a superfluity of those points throughout the model domain and then ask PEST to find for itself those regions within the study area where transmissivity must be greater or less than average in order to ensure that there is good agreement between model outputs and field measurements. If we had based our parameterization of the model domain solely on zones, we might not have placed those zones in the correct position for the calibration process to properly infer the existence or extent of such heterogeneity.

The introduction of regularization into the calibration process serves two purposes. Firstly it brings a high degree of numerical stability to a parameter estimation problem which would otherwise be highly susceptible to the deleterious effects of a singular normal matrix (you might have noticed when inspecting hcal.rec that PEST was not able to calculate any parameter statistics due to singularity of the normal matrix.) Secondly, if regularization constraints are appropriately defined, model calibration can proceed with a “homogeneous unless proven otherwise” philosophy; that is, in spite of the number of parameters at its disposal, PEST will make each zone within the model domain as uniform as it can in terms of the distribution of the estimated hydraulic property, introducing heterogeneity into a zone only where this is necessary in order to allow a good of fit between model outputs and field data to be achieved. Hence any heterogeneity which is introduced as an outcome of the calibration process is “there because it has to be there”. In many modeling contexts this philosophy of model calibration has a large intuitive appeal, allowing a modeler to use zones to characterize the distribution of some hydraulic property within a model domain while, at the same time, removing the inflexibility that accompanies the characterization of a model domain by areas of piecewise parameter constancy.
Relationships between pairs of parameter values can be introduced into the calibration process as prior information equations. The weight assigned to each of these prior information equations can be the same. Alternatively, if the weight is proportional to the inverse of the square root of the variogram calculated for the distance between the respective pilot points, then it can be shown that this is in harmony with the geostatistical characterization of the area as encapsulated in the variogram. What this characterization says, in short, is that “the closer are two points together, the more likely are the hydraulic properties at those points to be the same”. By calculating weights on the basis of the inverse of the variogram, we are enforcing the “zero difference” condition more strongly for points which are closer together than for those which are farther apart.

When run in this mode, a number of control variables are required in the PEST control file, in addition to those required when PEST is run in “parameter estimation” mode. One of these variables is PHIMLIM. This specifies the degree of model-to-measurement misfit that is allowed to occur in the present optimization process. Because the attainment of a good model-to-measurement fit, and the simultaneous enforcement of homogeneity constraints, may place conflicting requirements on parameter values, a compromise between the two must be reached. The user determines the “compromise level” by setting a maximum model-to-measurement misfit that he/she will tolerate, this misfit being expressed in terms of the “measurement objective function”. The maximum permissible value of the measurement objective function (i.e. PHIMLIM) should be set a little higher than the objective function than it is possible to achieve without any regularization constraints being enforced.

Each prior information equation included in the parameter estimation process must be assigned a weight. As was discussed above, weights are calculated on the basis of geostatistical information available (or assumed) for the model area. If an observation or prior information equation is used for regularization purposes, then it is assigned to the observation group “regul”. As part of its regularization functionality, PEST adjusts the weights assigned to all members of this group during each iteration of the optimization process; however the relative weight values within this group remain the same. The “regularization weight factor” by which the initial weights of all members of the group “regul” are multiplied during each optimization iteration is calculated in such a way as to respect the PHIMLIM value provided by the user as the maximum tolerable model-to-measurement misfit for the current case. An initial regularization weight factor needs to be supplied by the user.

13.5.2.1 No Regularization

If you choose the No Regularization option, then no further settings need to be adjusted prior to the PEST run. You will be presented with the PEST Control file, whereby you can edit/adjust values such as the PEST objective function goal, number of iterations, etc. Please consult the PEST users manual for an explanation of these parameters.
13.5.2.2 Tikhonov Regularization

When you select Tikhonov Regularization, the PEST control file will automatically be generated to support this feature.

Select the [Generate Tikhonov Regularization] button to select the type of regularization.

- **Preferred Homogenous (Smoothness)**: Prior information equations relate the pilot points to one another. Pilot points that are near one another should have close values; only enough heterogeneity will be introduced to the system as necessary, the level of the heterogeneity being controlled by the pertaining semi-variogram.

- **Preferred Value**: Prior information equations relate the pilot points to their initial value. PEST will try to find a value that is as close to the initial pilot point value as possible.
When you are finished, proceed to the [Run PEST] step to begin the PEST run.

### 13.5.2.3 SVD Assist

#### Theory

In contrast to Tikhonov regularization, which adds information to the calibration process in order to achieve numerical stability, subspace methods achieve numerical stability through subtracting parameters, and/or parameter combinations, from the calibration process (Aster and others, 2005). As a result of the subtraction, the calibration process is no longer required to estimate either individual parameters or combinations of correlated parameters that are inestimable on the basis of the calibration dataset. These combinations are automatically determined through singular value decomposition (SVD) of the weighted Jacobian matrix (Approaches to Highly Parameterized Inversion: a Guide to Using PEST for Groundwater Model Calibration: [http://pubs.usgs.gov/sir/2010/5169/](http://pubs.usgs.gov/sir/2010/5169/)).

When PEST implements “SVD-assisted” parameter estimation, it computes the global Jacobian matrix only once. Then it decomposes parameter space into estimable and inestimable parameter combinations. It then reformulates the whole calibration problem so that, from that moment on, it estimates only “super parameters”. Only as many of these are required as there are dimensions in the calibration solution space; alternatively, only as many
of these need to be defined as you have computing resources to estimate. Through the use of super parameters you can get all of the benefits of highly parameterized inversion with a comparatively small run time burden. Your model can have hundreds, or even thousands, of parameters, but these may be accommodated with a computational burden of only a few tens of runs per iteration.

Click on the [Create SVD Assist Run] button on the toolbar, as shown below.

The following Settings dialog will appear. For SVD Assist, you must provide the number of Super Parameters to use. Please consult the PEST manual for a recommended set of parameter values.
After clicking OK in the settings window, the PEST Control file, customized for an SVD Assist run, will be populated as shown below.
When you are finished, proceed to the [Run PEST] step to begin the PEST run.

13.6 Run PEST

At this step, you can run PEST Check and start/stop the PEST run.
Before launching the PEST run, it is a good idea to run PEST Check. This PEST utility will check all the input files to ensure they meet a minimum set of criteria.

Click on the **PEST Check** button on the toolbar. If there are errors, you will receive a notification. (if there are just warnings, you will not receive a notification)

Click the Run PEST button to start the pest.exe. The progress should appear in a DOS window.

Once PEST finishes, you should see a confirmation message appear under the PEST Run Log tab, indicating if the PEST run was successful or not.

Click (Next Step) to proceed to the **Analyze Results** step.
13.7 Analyze Results

After the PEST run is successful, you can analyze the results; Visual MODFLOW Flex provides a link to the PEST output files:

- Record file (.REC)
- Sensitivities for Observations (.SEO)
- Sensitivities for Parameters (.SEN)
- Residuals (.RES)

The Analyze Results workflow window includes several tabs which display the output files from the PEST run (i.e. .REC, .SEO, .SEN and .RES files).

The .REC tab displays the **Record File** for the PEST run. The record file includes detailed record of the entire parameter estimation process, listing the number of parameters,
parameter groups, observations, parameter definitions/settings, control settings, initial conditions, etc. The record file also lists the parameter values for each iteration of the model.

The .SEO tab displays the Observation Sensitivity file. This file records the sensitivity of each individual observation with respect to all parameters involved in the PEST run. Observed and modeled values are listed for each observation point/time, along with the composite sensitivity of each observation point/time with respect to the estimated parameters.

The .SEN tab displays the Parameter Sensitivity file. This file records the sensitivity of each individual parameter with respect to all observations involved in the PEST run. The parameter value and sensitivity are listed for each iteration of the PEST run.

Finally, the .RES tab displays the Residual file. This file displays the measured and modeled values for each observation point based on the BEST results achieved by PEST (i.e. the model iteration with the lowest Φ (phi) value. In addition, the residual value for each observation is displayed.

The PEST output files can be exported to Excel spreadsheets for further processing and charting. Click on the button to specify an output name/location for the Excel file; the results will be separated into individual worksheets for facilitate analysis.

13.8 Save PEST Parameters as New Inputs

At the end of a PEST run (other than a Sensitivity Analysis), you may want to save the final parameters as inputs for a new model run, provided the adjusted parameters are reasonable.
Click on the "Update Model with PEST Results" button.

Visual MODFLOW Flex will save the adjusted model parameters in a new model run within the same project. This new model run will appear in the Model Explorer below the most recent model run. You must Translate and Run this new model run in order to see the updated MODFLOW results.

In the past, you would need to do a "Save Project As...", if you wanted to keep the original model separate from the PEST updated model. With Visual MODFLOW Flex, you can have all these model scenarios/runs accessible within a single project, allowing you to efficiently compare the original model to the "PEST-adjusted" model run, where you can make comparisons between calculated heads, property zone parameters, etc.

13.9 Making Changes after a PEST Run

After PEST has been run to completion, you may need to make changes to the defined PEST inputs, and/or the numerical model. Please follow the steps below in the case of these changes:

**Numerical Model**

If you change the model engine, you must go back to the start of the PEST workflow, and click Apply and Next at each step right up to the Run PEST step.

If you change the Translation settings (e.g. Solver, Package settings) and or the property package (LPF vs BCF), just go back to the Define Kriging parameters step, click [Apply] button, then you can go directly to Run PEST.
If you need to change the parameter zonation (e.g. add or remove Conductivity zones), this is something that is not yet updated in the PEST workflow. Please make a clone of the model run, change the parameter zonation in this clone, then launch a PEST run from this model clone.

PEST Inputs

If you change the Observations and/or Weights:
- Return to this step in the PEST workflow and make the necessary changes
- Click [Apply] button at the top of the workflow toolbar
- Go to the Run PEST step

If you change the Property Zone parameter min/max values
- Make the changes at this step in the workflow
- Click [Apply] button
- Go to Define Kriging Variograms, and click [Apply] to re-generate these PEST input files.
- Go to the desired regularization step, and Regenerate the PEST Control file.
- Go to the Run PEST step and Run PEST.

If you change the pilot point values, or add new pilot points
- Make the changes at this step in the workflow
- Click [Apply] button
- Go to Define Kriging Variograms, and click [Apply] to re-generate these values
- Go to the Run PEST step.
14 Working with Multiple Model Scenarios

One of the unique features of VMOD Flex is the ability to work with multiple numerical models inside a single project. Unlike other modeling GUI’s, VMOD Flex does not restrict you to a single model scenario or single grid for your model. With the workflow based approach, you can generate multiple grid realizations from your conceptual model, each one with one or more model runs. Likewise, you can import multiple MODFLOW models or Visual MODFLOW projects into a single project, and analyze and compare these projects.

Loading a New Workflow

There are a number of ways in which a new numerical model workflow window will be created:

- You have imported a MODFLOW or Visual MODFLOW project, immediately after creating a new project.
- You have imported an additional MODFLOW or Visual MODFLOW model into an existing project. To use this option, select the Workflow/Numerical Model menu option.
- You have “Duplicated” or Cloned an existing numerical model, through the Clone option. To use this option, right click on the “Run” node on the model tree, and select “Clone”. When this option is selected, the model run tree in the model explorer will be duplicated, and a new workflow window will appear.
- When you are within the Conceptual Model workflow, upon running the Conceptual to Numerical Model conversion.

Each new workflow will load as a separate window inside the VMODFlex main application window.

The windows can be accessed from the window toolbar at the top of the screen, as highlighted in red below.

The tree structure for multiple model runs is shown below. Each numerical grid (which is derived from a conceptual model) can have one or more model Runs; each model run contains the inputs and output for a single numerical model.
Closing/Minimizing Workflow Windows

You can close a workflow window at anytime by selecting the X button in the top right; however, this is only recommended if you do not need the workflow window until a later time (for example a specific model run).

How To Reload a Closed Workflow

You can reload a workflow window at any time from the Workflow menu, by selecting Workflow/Open, and the following dialog will appear:
Select the desired workflow(s) and click [OK].

In addition, you can load a specific workflow through the model.

To reload a Conceptual Model workflow, right click on the Conceptual Model node in the model tree, and select Open Related Workflows...

To reload a Numerical Model workflow, right click on the desired **Run** node in the model explorer, and select Open Related Workflow...
15 VMOD Flex and VMOD Classic

Introduction

This document describes how to pass models from Visual MODFLOW Flex to the Visual MODFLOW Classic Interface and how to load the results from a Classic run back into the Flex environment.

When do you need to switch to VMOD Classic Interface?

All new projects should begin within the Flex interface; however, if you need to use the following features, then you will need to migrate to the Classic interface to complete that portion of your project:

- Flow Engines (MODFLOW-96)
- Flow Packages (ETS1, STR, MNW)
- Transport Engines (MT3D99, PHT3D)
- MGO

Visual MODFLOW Flex

All new projects should begin within the Flex interface.

Create Your Conceptual Model

The first step is to create your conceptual model using the Visual MODFLOW Flex interface. Starting with a conceptual model can save you a significant amount of your total project time, because the material properties and hydrological boundaries are done independent of the grid design and numerical engine selection. This enables rapid adaptation of grid requirements to match property and boundary definitions, thus reducing modeling instability and convergence problems.

Here are a few topics to get you started:

- Conceptual Modeling Tutorial
- Conceptual Modeling Workflow

Some sample images from the conceptual model demo project are below:
Figure 1: Property Zones in the Conceptual Model

Figure 2: Boundary Conditions in the conceptual model: (constant heads on the east and west boundaries (colored red); a river through the middle of the model (colored blue), and 4 extraction wells)
When you transfer your flow model to Visual MODFLOW Classic, the names for the boundary conditions and wells are not preserved. If this is important, define only the geological model in Visual MODFLOW Flex, then add the boundary conditions in Classic.

**Generate MODFLOW Grids**

Once the conceptual model is designed, you must generate one or more MODFLOW grids. This process is described in Conceptual Modeling Tutorial and further explained in the help section: “Creating MODFLOW Grids”

Visual MODFLOW Flex allows you to generate various numerical representations of your conceptual model; this means you can experiment with various grid sizes, horizontal and vertical discretizations, etc. This includes grids with Local Grid Refinement (LGR).

After a grid is generated, you may see something similar to the image shown below (in a 2D plan view).

*Figure 3: Numerical Grid with Conceptual Boundary Condition Objects*
Convert Conceptual Model to Numerical Model

After the grid is designed, you must populate the numerical model. Run the Conceptual to Numerical Conversion wizard as shown below:

This will generate a numerical model with grid layers, property cell representations, and set of cells corresponding to the boundary conditions you created in the conceptual model. This process is also described in the help section: “Converting Conceptual to Numerical Models”

💡 VMOD Flex allows you to generate various numerical models from your conceptual model. You can calculate the difference in heads or drawdown between several model runs.
For more details, see “Compare Heads and Drawdown”

Once the numerical model is created, you will see something similar to the examples below:
Figure 4: Property Zones in the Numerical Workflow

Figure 5: Property Zones in a stand-alone 3D Viewer
Translate to MODFLOW Packages

After reviewing the numerical model, you are ready to generate MODFLOW packages that will serve as inputs for the MODFLOW 2000/2005 Run. At the Translation step in the numerical workflow, click on the Translate button.
Make a note of the **Output Folder** (circled above) for your project. This is where VMOD Flex will generate the MODFLOW files on your hard drive. These are the files you will need to select when you import into Visual MODFLOW Classic as explained below.

If you are running a flow model, at this point you could proceed to the Run Engines step in VMOD Flex. For more details, see “**Run Numerical Engines**”

**VMOD Classic**

Change over to the Visual MODFLOW Classic interface in order to add and complete the following portions of your project:

- Flow Engines: MODFLOW-96
- Flow Packages: ETS1, MNW, and/or STR
- Transport Engines: MT3D99, PHT3D
- MGO

**Import MODFLOW Files (from VMOD Flex)**

Visual MODFLOW Flex translates the numerical models into standard MODFLOW package files. To migrate your model from Flex to the Classic interface, you must import these translated files into Classic as described below:

1. Start Visual MODFLOW Classic
2. Select File / Import MODFLOW.
3. Browse to the folder that contains the Translated MODFLOW files from VMOD Flex (the Output Folder circled above)
4. Choose the project_name.MODFLOW.IN file that was generated by VMOD Flex.
5. Enter a name for the Visual MODFLOW project.
6. Follow the Import MODFLOW Wizard; if you need assistance, refer to the Import MODFLOW Files section in the VMOD Classic web help.

After the model is imported, you can review the model inputs in the VMOD Classic Interface.
Figure 7: Property Zone Database in VMOD Classic
Add Transport Inputs or Pathlines

Use Visual MODFLOW Classic to add inputs for Transport (or Particles for Pathlines), then translate and run the model. For more information on these options, refer to the Visual MODFLOW Demo Tutorial or the help documentation.

- Tutorials: https://www.waterloohydrogeologic.com/visual-modflow-flex-tutorials/
- Web help for VMOD Classic: https://www.waterloohydrogeologic.com/help/vmod/

View Results

After running the model in VMOD Classic, with MT3DMS or MODPATH, the results will appear something similar to below:
Figure 9: Pathlines in VMOD Classic
Export Results

The VMOD Classic interface supports several different export file formats. Model results from the VMOD Classic interface that are exported as shapefiles or 3D gridded data objects can be easily imported into the VMOD Flex interface for superb 3D display and comparison with initial project conceptualizations and raw data objects.

Shapefiles

Once the models are completed in VMOD Classic, you can export the results in several ways.

- Concentration contours can be exported as contour lines (File / Export GIS, and choose Contours Shapefile). This must be done on a per layer basis, and for each desired output time.
- Pathlines can be exported to .SHP file. (File / Export GIS, and choose Contours Shapefile.)
- These files will be imported as Polyline Data Objects in VMOD Flex
3D Gridded Data Objects - Tecplot (.DAT)
Concentration results can also be exported to a TecPLOT .DAT file, which contains concentration values for all cells in all layers. The benefit of this format is that you can create cross-sections of concentration profiles and also isosurfaces (3D Plumes) in VMOD Flex.

- Select File / Export / Data and choose TecPLOT .DAT file format. This must be done for each desired output time.
- This file will then be imported as a 3D Gridded Data object in VMOD Flex, as explained below.

Visual MODFLOW Flex

Import Results from VMOD Classic

Polylines (Shapefiles)
In VMOD Flex, select File / Import and choose Polyline as the data type. Select the desired pathline or concentration contour polyline shapefiles. The polylines will not have any elevation attribute. If you wish to position these in a 3D Viewer, you need to define an elevation. A simple way to do this is through data operations. Select the desired polyline and load the Settings / Data Operations. Select a desired Z (Elevation) value or select to have the Z value calculated from a surface. Execute the Data operation and click OK to update the data object. For more details, see “Data Operations” in the web help.

The following figures illustrate the pathlines and contour lines visualized in 3D viewer, after assigning an elevation attribute to each polyline data object.
Figure 11: Pathlines in 3D in VMOD Flex with Pumping Wells and Horizons visible
Polyline data objects may also be added on the Layer view of the Numerical model, as shown below.
Concentrations as 3D Gridded Data Object (TecPlot .DAT)

The Concentration results from VMOD Classic can also be imported as a 3D Gridded Data objects; this format provides a richer set of visualization options, such as 3D Isosurface, colormap along specific row, column, or layer, or arbitrary cross-section line. In order to use this option, you must export your concentration results to a TecPot .DAT file as explained above.

Select File / Import and choose 3D Gridded Data as the object type. Select the .DAT file that was generated from VMOD Classic and proceed through the default import settings. After the file is imported, visualize in a 3D Viewer. Refer to “Style Settings – 3D Gridded Data” in the webhelp for assistance in setting up color maps or isosurfaces.
Figure 14: Concentrations as 3D Grid, rendered as an Isosurface; Fence Diagrams represent the conductivity zones
For More Information

If you require assistance during importing of models into VMOD Classic, or the results back into VMOD Flex, please contact our Tech Support team:

support@waterloohydrogeologic.com.
16 References

Online Guides to USGS MODFLOW and associated packages:


References


Appendix A - Input files and Packages

When you "Translate" in the numerical model, a number of text files will be created; these input files (called packages) are required for the numerical models (engines) to run. These files are saved in the "Run" directory for the current grid/run that you have selected, in the data repository; for example:

\Drumco.data\MODFLOW\NumericalGrid1\Run\MODFLOW-NWT\..

If you choose a different engine (at the "Select Engines" step), then a new folder will be created under that run.

In Visual MODFLOW Flex, most of the packages containing arrays will have a main "index" file which lists global settings/parameters and provides file references to the other files that are also inputs for that package. An example is the MODFLOW .DIS package; some general parameters (such as units) and temporal discretization (number of stress periods and time steps) are described in this projectname.DIS file; however, cell elevations for each layer are described in their own separate file, for example Projectname.DIS.Bot1 (contains bottom elevations for each cell, for layer 1)

A brief description of these files is listed below, where projectname is the name assigned to the model.

MODFLOW (2000, 2005, NWT)

A brief description of these files is listed below, where projectname is the name assigned to the model.

Input files:

- projectname.modflow.IN List of translated files for MODFLOW
- projectname.BAS: Basic Package data file
- projectname.BCF: Block-Centered Flow Package data file.
- projectname.CHD: Time-variant specified head data file
- projectname.DIS: Model discretization data file
- projectname.DRN: Drain Package data file
- projectname.EVP: Evapotranspiration Package data file (MODFLOW-96).
- projectname.GHB: General-Head Boundary Package data file
- projectname.LPF: Layer Property Flow data file
- projectname.OC: Output control options data file
- projectname.PCG: PCG2 Solver Package data file
- projectname.RCH: Recharge Package data file
- projectname.RIV: River Package data file
* projectname.SIP: SIP Solver Package data file
* projectname.SOR: SOR Solver Package data file
* projectname.UZF: Unsaturated Zone Flow Package data file
* projectname.WEL: Well Package data file
* projectname.WHS: WHS Solver data file

**Packages specific to MODFLOW-NWT:**
* projectname.UPW: Upstream Weighting (property package, only when MODFLOW-NWT is selected)
* projectname.NWT: NWT Solver settings (this solver must be used when MODFLOW-NWT is selected)

**Packages specific to MODFLOW-USG:**
* projectname.CLN: connected linear network package file
* projectname.GNC: Ghost-node correction package, which includes additional node coordinate information facilitate corrections for cell geometries that do not inherently satisfy the requirements of the flow formulation in MODFLOW-USG
* projectname.GSF: Grid Specification file, which includes grid cell vertex coordinates and connectivity
* projectname.SMS: Sparse Matrix Solver package file

**Output files:**
* projectname.BGT*: MODFLOW file containing water budget data used by MODPATH
* projectname.CBB*: MODFLOW-SURFACT file containing water budget data used by MODPATH (Binary format)
* projectname.CBC*: MODFLOW-USG file containing water budget data used by MODPATH3DU (Binary format)
* projectname.DDN: MODFLOW file containing drawdown X, Y, Z heads for each node (Binary format)
* projectname.FLO*: MODFLOW file containing cell-by-cell flow terms (Binary format)
* projectname.HDS*: MODFLOW file containing head results (Binary format)
* projectname.LST*: MODFLOW listing file containing information and messages from MODFLOW (ASCII format)

**MODPATH**

**Input files:**
* BACKWARD.IN: Backward tracking particle locations data file
* BACKWARD.PTH: Backward tracking run options data file
* FORWARD.IN: Forward tracking particle locations data file
* FORWARD.PTH: Forward tracking run options data file
- projectname.MPT Main MODPATH input data file
- MODPATH.IN List of MODPATH input files

**Output files:**
- projectname.CBF Composite Budget file used by MODPATH
- projectname.MPB MODPATH file containing backward tracking particle information (ASCII format)
- projectname.MPF MODPATH file containing forward particle information (ASCII format)
- SUMMARY.PTH* MODPATH file containing listing information and messages from MODPATH (ASCII format)

**MOD-PATH3DU**

**Input files:**
- Conceptual Model.JSON Primary input file in JavaScript Object Notation (JSON) format
- Conceptual Model.MP3DU.GSF grid specification file compatible with MOD-PATH3DU
- Conceptual Model.MP3DU.IN model input file
- Conceptual Model.P3D model property file
- One set of files per input particle group:
  - Conceptual Model_{pathiline group}.shp shapefile with particle group starting location coordinates
  - Conceptual Model_{pathiline group}.dbf shapefile database with particle group attributes
  - Conceptual Model_{pathiline group}.shx shapefile spatial index

**Output files:**
- MP3DU.log output listing file in JSON format
- One set of files per corresponding output pathline group:
  - Conceptual Model_{(globally_unique_id)}.PATHLINE.BIN binary output file with particle group pathlines
  - Conceptual Model_{(globally_unique_id)}.PATHLINE.shp shapefile with particle group pathlines
  - Conceptual Model_{(globally_unique_id)}.PATHLINE.dbf shapefile database with pathline attributes
  - Conceptual Model_{(globally_unique_id)}.PATHLINE.shx shapefile spatial index

**MT3DMS / RT3D / SEAWAT**

**Input files:**
- projectname.AD3 Advection data file
● projectname.BT3 Basic Transport data file
● projectname.DP3 Dispersion data file
● projectname.GCG Transport Solver data file
● projectname.RC3 Chemical Reaction data file
● projectname.SS3 Source / Sink Mixing data file
● RT3D.IN List of RT3D input files

**Packages specific to MT3DMS:**
- MT3D.IN List of MT3D input files

**Packages specific to RT3D:**
- RT3D.IN List of RT3D input files

**Packages specific to SEAWAT:**
- SEAWAT.IN List of SEAWAT input files
- projectname.VDF Variable Density Flow data file
- projectname.VSC Viscosity data file

**Output files:**
- projectname.OT* MT3D output file containing listing information and messages from MT3D e.g. cumulative mass budget data (ASCII format)
- projectname.CNF* MT3D file containing model grid configuration file (ASCII format)
- projectname.MAS* MT3D file containing mass balance file (ASCII format)
**Please Note:** The mass balance files are displayed by MS-Windows as projectname.001, projectname.002, etc.
- projectname.OBS* MT3D file containing concentration at observation points (ASCII format). The data is presented with units of mass/length^3
- projectname.UCN* MT3D file containing unformatted concentration information - (Binary format)
- projectname.CBM/CCM* MT3D96/MT3D99 cell-by-cell mass terms (Binary format)

**ZONEBUDGET**

**Input files:**
- projectname.ZBI Zone Budget data file
- ZONEBUD.IN List of Zone Budget input files

**Output files:**
- projectname.ZOT Zone Budget file containing water balance data (ASCII format).
Appendix B - VM Flex GUI and Input Files

MODFLOW (2000, 2005, NWT)
A brief description of these files is listed below

**BAS: Basic Package**
Array data originates from various spots in the Input:
- View/Edit Grid (Active/Inactive cells)
- Properties / Initial Heads
- Define Boundary Condition (Constant Heads only, steady state only)

Some settings are defined at Translate step, check the Initial Heads option.

Reference files in this package may include the following external array files (one each per layer, with the digit suffix representing the layer index):

OPEN/CLOSE Name.BAS.ibound1
Active flag for each cell: active, constant head, or inactive

OPEN/CLOSE Name.BAS.head1
Initial head value for each cell; the values will depend on what option you chose for Initial Heads at the Translate step.

**DIS: Discretization Package**
Contains the spatial and temporal discretization; also contains the units. Array data originates from Input, View/Grid (# rows, cols, layers, layer top, layer bottom). Some settings are defined at Translate step, Time Steps. Units are defined in Proj settings. # Stress periods is based on the boundary conditions in the input.
- The first sets of arrays contains the cell width.
- The second set of arrays contains the cell height.
  Note: These only need to be defined once, since all layers in a finite difference model must have same cell width and height

After that are links to the cell elevation for each layer.

OPEN/CLOSE Name.DIS.Top' 1 (FREE) -1  #Top Elevation
The cell top elevation value for cells in layer 1

OPEN/CLOSE Name.DIS.Bot1' 1 (FREE) -1  #Bottom Elevation of layer 1
The cell bottom elevation value for layer 1

**Property Packages**

**BCF: Block Center Flow package**
Used for MODFLOW-2000 and 2005
Array data originates from Input/Properties. Conductivity and Storage.
Some additional settings are defined at Translate step, mainly using the Layer types, Rewetting, and Anisotropy options.

Details on the Referenced files from within this package are explained below:

Starting at line 5:

OPEN/CLOSE Name.BCF.ss1' 1 (FREE) 0 #Ss (specific storage) for layer 1
(this file will contain cell-by-cell values for specific storage, for layer 1)

OPEN/CLOSE Name.BCF.hk1' 1 (FREE) -1 #HK for layer 1
(this file will contain cell-by-cell values hydraulic conductivity values for layer 1)

OPEN/CLOSE Name.BCF.vcont1' 1 (FREE) 0 #VCONT for layer 1
(this file will contain cell-by-cell values VCONT values for layer 1)

If you have enabled additional options such as Anisotropy "As Specified" or Cell Re-wetting, you may see references to additional parameters.

**LPF: Layer Property Flow Package**
Same as BCF; the main difference is that Kz is explicitly defined (there are no values for VCONT or Transmissivity)

OPEN/CLOSE 'Conceptual Model 1.LPF.hk1' 1 (FREE) -1 #HK for layer 1
(this file will contain cell-by-cell values for Horizontal Conductivity (Kx) for layer 1)

OPEN/CLOSE 'Conceptual Model 1.LPF.vka1' 1 (FREE) -1 #VK for layer 1
(this file will contain cell-by-cell values for Vertical Conductivity (Kz) for layer 1)

OPEN/CLOSE 'Conceptual Model 1.LPF.wetdry1' 1 (FREE) -1 #Wetdry for layer 1
(this file will contain wet dry flags for layer 1)

**UPW: Upstream Weighting Package**
Same format as LPF package.

**Boundary Conditions**
These are specified at the "Define Boundary Conditions" step in the numerical modeling workflow.

The following boundary conditions use format "Layer, Row, Column, parameter, ...." in their package (and do not use arrays)

- Constant Head
- River
- General Head
- Drain
- Wall (HFB)
- Specified Flux
- Wells
The format of these packages is described in the USGS MODFLOW documentation. The remaining boundary conditions in VMOD Flex require defining values in one or more arrays. These are described below.

**Recharge (RCH)**
Array data originates from Input, Define Boundary Conditions/Recharge and EVT. Some settings are defined at Translate step, Recharge.

- Name.RCH.recharge0' 1 (FREE) -1  #Recharge
  Contains cell-by-cell recharge rate for stress period 1.

- Name.RCH.recharge1' 1 (FREE) -1  #Recharge
  Contains cell-by-cell recharge rate for stress period 2.

***This is repeated for subsequent stress periods for a transient simulation
Comment: need a test project that distinguishes between top layer and topmost active layer

**Evapotranspiration (EVT)**
Array data originates from Input, Define Boundary Conditions/Evapotranspiration
Some settings are defined at Translate step, Recharge and EVT.
In the input (click on database), you define the following parameters:
- Evapotranspiration Rate
- Extinction Depth

- Name.EVT.elevation' 1 (FREE) -1  #Elevation
  Contains the ground elevation values for layer 1

- Name.EVT.evapotranspiration0' 1 (FREE) -1  #Evapotranspiration
  Contains the evapotranspiration rate for each cell, for stress period 1.
  ***This is repeated for subsequent stress periods for a transient simulation

- Name.EVT.depth0' 1 (FREE) -1  #Extinction Depth
  Contains the extinction depth values for each cell, for stress period 1
  ***This is repeated for subsequent stress periods for a transient simulation

**Lake (LAK)**
Array data originates from Input, Define Boundary Conditions/Lakes
Some settings are defined at Translate step, "Lakes"

- Name.LAK.lkarr1' 1 (FREE) -1
  Active/inactive cells; for any cell that is defined with a lake, it must be marked as inactive.

- Name.LAK.bdlknc1’ 1 (FREE) -1  #Lake bed leakance for layer 1
  Lake bed leakance for each cell, for layer 1. This is calculated from the parameters defined for the lake at the Boundary conditions step.
ZONEBUDGET

.ZBI contains the array of cells, and what zone ID is assigned to each cell. All layers are contained within a single file.

MODPATH

.MPT file is the main input file; it contains some general information at the top.

Details on the Referenced files from within this package are explained below:

OPEN/CLOSE 'Conceptual Model 1.MPT.ibound1' 1 (FREE) -1
Active/inactive flag for cells, for layer 1. (the values are the same as those in the .BAS package)

OPEN/CLOSE 'Conceptual Model 1.MPT.ep1' 1 (FREE) -1 #Ep (Effective Porosity) for layer 1
Effective porosity values for each cell, for layer 1. These values originate from the Define Properties step, "Storage" parameters.

MT3DMS / RT3D / SEAWAT

AD3 Advection
Advection scheme and settings; defined at the Translation step, MT3DMS Translation settings

BT3 (Basic Transport)
Contents of this file:

- Cell column data (cell width): this is based on the cell dimensions. There is just one array for all layers in the model
- Cell row delta (cell height): same as above
- Htop/water table elevation: Top elevation of the cell, or the water table elevation, whichever one is higher. There is just one array.

You can find this by looking at the cell dimensions at the Input/Grid step
- Layer Thickness: one array per layer. This is based on the cell height. You can see this at the view/edit grid step.
- Porosity: one array per layer. Input for this is defined at Define Properties/Storage. (note: Total [default] or effective porosity can be selected here)
- icbund: active/inactive cells for layer 1. (1=active, 0=inactive)
- Starting Conc. (S Conc) - Layer 1: at Define Properties/Initial Concentration step.

Also contains the "saved" output times for the MT3DMS run; these are defined at the Translate step, under MT3DMS node.
**DP3 (Dispersion)**
Contains the Dispersion parameters; these are defined at the Define / Properties step, Long. Dispersion.

**RC3 (Chemical Reactions)**
Contains the chemical reaction parameters; these are defined in Flex as "Species Parameters" and "Model Parameters". Default values and some global parameters are defined at the Define Modeling Objectives step; cell-by-cell values for model parameters and species parameters are defined/assigned/edited at the Define Properties step.

**SS3 (Sinks / Sources)**
Contains the Sink/sources (transport boundary conditions). These are defined at the "Define Boundary Conditions" step; these can be Constant Concentration, concentrations assigned to Pumping Well rates, or can be species concentrations for existing boundary conditions such as River, Recharge, EVT.

**GCG Solver**
Solver parameters and time steps; defined at the Translation step, MT3DMS/RT3D/SEAWAT Translation settings

**SEAWAT Only**

**VDF (Variable Density Flow)**
Contains the variable density settings and parameters; some of these parameters are defined at the Define Modeling Objectives step; settings and a few relevant parameters are set using the advanced Transport Engine settings at the Translation step.

**VSC (Viscosity)**
Contains the viscosity package settings and parameters; some of these parameters are defined at the Define Modeling Objectives step; cell-by-cell values are defined at the Define Properties step; settings and a few relevant parameters are set using the advanced Transport Engine settings at the Translation step.
Index

- 3 -

3D Record to AVI File 355
3D View
Settings 40
3D Viewer Performance Preferences 37

- A -

About the Import Process 280

- C -

Cell Inspector 351
  cell values 352
  select items 352
Concentrations 705
Constant Head 464
Coordinate System 44
Cross Sections
  Plot on Color Map 395

- D -

Decaying Source 554
Define Observation Wells 628
Define Particles 636
  MODPATH 636
  mod-PATH3DU 636
  Using Circle 639
  Using Points 638
  Using Points Data Object 642
  Using Polygon 641
  Using Polyline 640
Define ZoneBudget Zones
  Point 633
  Polygon 633
  Polyline 632
  Using Data Object 633
Define ZoneBudget-USG Zones
  Point 758
  Polygon 758
  Polyline 757
  Using Data Object 758
Defining V-Grid
  Feature Refinement 522
  Finish 524
  Grid Size 521
  Layer Refinement 522
  Preview 523
  Toolbar 521
Delete Particles 643
Display Settings MODFLOW-USG Properties 747

- E -

Export
  Contour Lines 742
  Pathlines 742
Export Charts 728
Export views to image 358
Expression Builder 787
  data objects 791
  functions 791
  horizons 791
  operators 788
  surfaces 791

- F -

Flex Viewer
  Settings 40
Flow Properties
  Edit Colors 576
  Edit Contours 576

- G -

General Settings 43

- H -

How to buy 30

- I -

Import Data 280
Maps 320
Import Data 280
  Points 280
  Surfaces 296
  Wells 304
Inactive Cells 568

- L -
Lake Package 663
Layer Type 665

- M -
Maps
  adding georeference points 324
  editing georeference points 324
  georeferencing images 321
Model Properties (Finite Difference)
  assign 572
  copy 575
  edit 574
MODPATH 636, 680
MOD-PATH3DU 636, 781

- N -
Numerical Grid
  Edit 105
  View 105
Numerical Properties
  Edit 83
  View 83

- O -
Observation Wells
  define wells 628
  observation groups 629, 754
Observations
  Adding Head Observation Data 365
OpenGL Driver Settings 41
Output Control Package 671, 776

- P -
Project Color Palette 37

- R -
Recharge Package 662
Rewetting 666
RT3D
  Decaying Source 554

- S -
Surfaces 388
  Colors 388
  Contour Lines 389

- T -
Translation Settings
  Advanced 673, 778
  anisotropy 670
  cell rewetting 666
  general 647
  initial heads 669
  lake package 663
  layer type 665
  MODFLOW-2000/-2005 647
  MODFLOW-LGR 650
  MODPATH 680
  MOD-PATH3DU 781
  MT3D-MS 682
  output control 671, 776
  recharge 662
  RT3D 682
  SEAWAT 682, 692
  solvers 651
  Solvers, GMG 653
  Solvers, NWT 660
  Solvers, PCG 651
  Solvers, SAMG 658
  Solvers, SIP 656
  Solvers, SOR 657
  Solvers, WHS 655

- U -
Units
  compound units 45
  consistent units 45
USG Translation Settings
  initial heads  774

- V -

Virtual Grid  41
Visualization
  Cells  391
  Colormap  394
  Cross sections  395
  Isolines  397
  Isosurfaces  399
  Points  375
  Slice  393
  ThreeSlice  401
  Time  400
  VelocityMap  403
  VelocityMap Color  405
  VelocityMap General  404
  Vertices  375

- W -

Wall (HFB) Boundary Condition  460