

# An Overview of MODFLOW FLEX

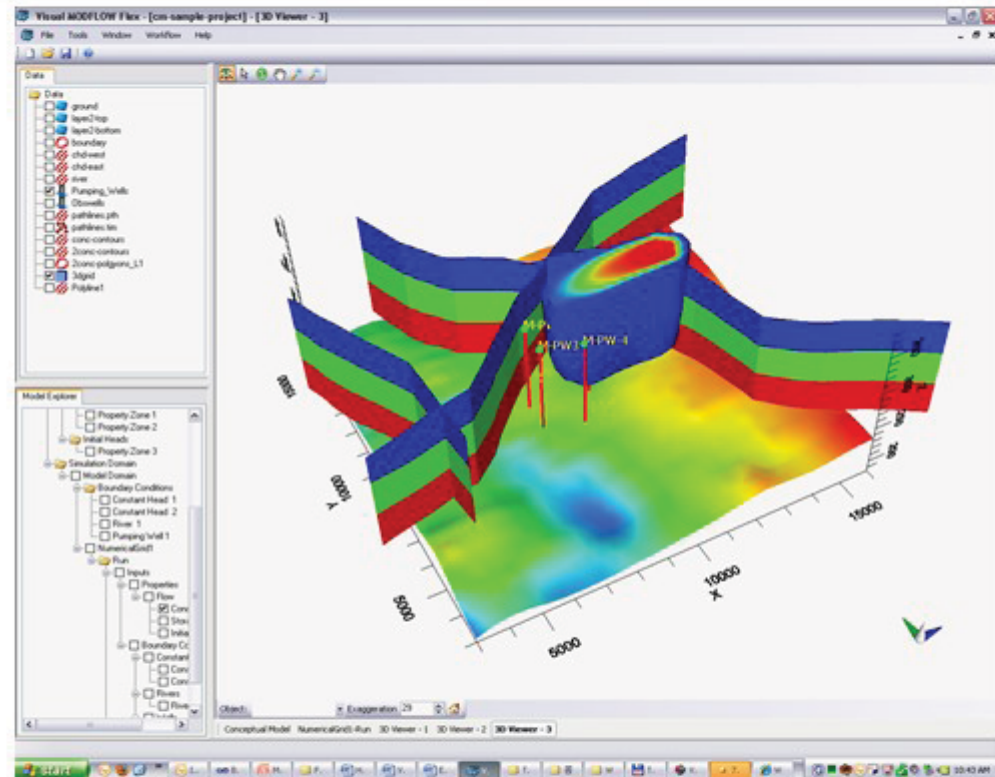
By:

**Dr. Surjeet Singh**

Scientist 'F'

Email: [surjeet.nihr@gov.in](mailto:surjeet.nihr@gov.in)

Mobile: +91-9456134747



**Ground Water Hydrology Division**

**National Institute of Hydrology**

Jalvigyan Bhawan, Roorkee-247 667

Uttarakhand (INDIA)



## **Presentation Overview**

- Modeling Process & Flow Equation**
- Visual MODFLOW FLEX**
- Comparison with Visual MODFLOW Classic**
- Additional Options in MODFLOW FLEX**
- Various Packages in FLEX**
- FLEX Overview**

## Modeling - Governing Equation

### Governing flow equation, PDE

$$\frac{\partial}{\partial x} \left( K_{xx} \frac{\partial h}{\partial x} \right) + \frac{\partial}{\partial y} \left( K_{yy} \frac{\partial h}{\partial y} \right) + \frac{\partial}{\partial z} \left( K_{zz} \frac{\partial h}{\partial z} \right) + W = S_s \frac{\partial h}{\partial t}$$

- $K_{xx}$ ,  $K_{yy}$  &  $K_{zz}$  are hydraulic conductivity along  $x$ ,  $y$ , and  $z$  coordinate axes (L/T)
- $h$  is the potentiometric head (L)
- $W$  is volumetric flux per unit volume representing sources and/or sinks of water, where *negative* values are extractions, and *positive* values are injections ( $T^{-1}$ )
- $S_s$  is the specific storage of porous material ( $L^{-1}$ )
- $t$  is time (T)

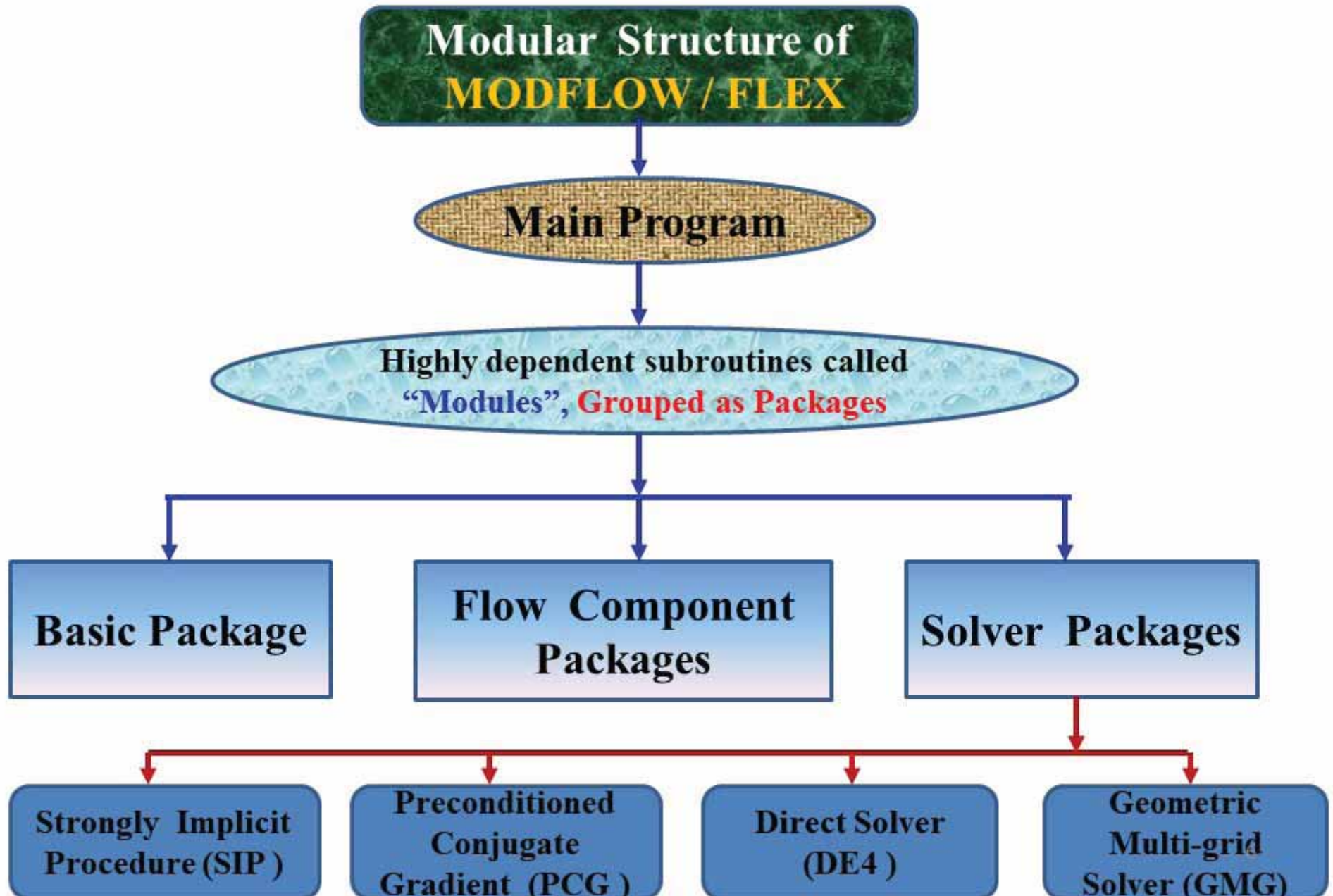
# MODFLOW

- **Developed by USGS (1984)**
  - A 3-D Finite Difference groundwater model;
  - Simulates and predicts groundwater conditions.
  
- **MODFLOW Packages include:**
  - Capabilities to simulate coupled groundwater/surface water systems, solute transport, variable-density flow (including saltwater), parameter estimation, and groundwater management.
  
- **MODFLOW-2005** – Most stable and well tested; simulates steady and non-steady flow in irregularly shaped flow system.  
( For both confined & unconfined or combined system)

## MODFLOW Releases ...

<b>MODFLOW-88</b>	Written in <a href="#">FORTRAN 77</a> , and its version 2.6, released in September, 1996.
<b>MODFLOW-96</b>	Released in December, 1996.
<b>MODFLOW-2000</b>	Released in July, 2000, and merged MODFLOWP and HYDMOD codes into the main program.
<b>MODFLOW-2005</b>	Standard version of MODFLOW.
<b>Visual MODFLOW</b>	Released in 1994 and based on the <a href="#">USGS MODFLOW-88</a> code and MODPATH code. Visual MODFLOW supports MODFLOW-2000, MODFLOW-2005, MODFLOW-NWT, MODFLOW-LGR, MODFLOW-SURFACT, and SEAWAT.
<b>Conduit Flow Process for MODFLOW-2005</b>	Simulates laminar and turbulent flow conditions within preferential flow layers and within single- and dual-porosity aquifers, as occurring in karst, fractured rock, and basalt aquifers.
<b>GSFLOW</b>	A coupled Groundwater and Surface water flow model based on the integration of MODFLOW-2005 with USGS Precipitation-Runoff Modeling System (PRMS).
<b>Visual MODFLOW Flex</b>	A graphical interface for MODFLOW. The program also combines proprietary extensions, such as MODFLOW-SURFACT, MT3DMS (mass-transport 3D multi-species) and a 3D model explorer.

# MODFLOW Packages



# Packages

Visual MODFLOW Flex has a no. of tools for analyzing water quality, groundwater supply, and source water protection.

<b>Tools</b>	<b>Features</b>
MODFLOW-2000, 2005, NWT	Standard Engine for groundwater flow modeling.
MODFLOW-USG	Finite volume version of MODFLOW with unstructured grids.
MODFLOW-LGR	Local grid refinement (LGR) for regional-local scale simulations.
MODFLOW-SURFACT	Saturated/ unsaturated subsurface flow and transport processes.
MT3DMS	Multi-species contaminant transport simulations.
SEAWAT	Variable-density groundwater flow coupled with multi-species solute and heat transport.
RT3D	Multi-species reactive contaminant transport simulations.
MODPATH	Standard package for forward and reverse particle tracking.
MOD-PATH3DU	Package for forward and reverse particle tracking supporting Un-Structured Grids.
Zone Budget	Package for sub-regional water budget calculations.
PEST v.12.3	Automated calibration and Sensitivity analysis tool.

# MODFLOW Engines

- **MODFLOW-2000, 2005, NWT:** Standard for groundwater flow simulations.
- **MODFLOW-USG:** Finite volume version of MODFLOW that uses unstructured grids.
- **MODFLOW-LGR:** Shared-node local grid refinement (LGR) for regional-local scale simulations.
- **MODFLOW-SURFACT:** Enhanced simulations of complex saturated/unsaturated subsurface flow and transport processes
- **MODPATH:** Standard package for forward and reverse particle tracking.
- **Zone Budget:** Standard package for sub-regional water budget calculations.
- **MGO\*:** For determining the optimal well pumping and/or injection rates at one or more wells, in order to achieve a specific objective while maintaining reasonable system responses.

\* *Available with the Classic interface.*



# **MODFLOW Packages**

**RIV** – River package

**CHD** – Constant-Head boundary package

**DRN** – Drain package

**WEL** – Well package

**LAK** – Lake Package

**EVT** – Evapotranspiration package

**STR\*** – Streamflow-Routing package

**RCH** – Recharge package

**FHB** – Flow and Head boundary package

**ETS1\*** – Evapotranspiration Segments package

**MNW\*** – Multi-node Well Package

**HFB** – Horizontal Flow Barrier Package

**UZF** – Unsaturated Zone Flow Package

*\* Available with the Classic interface.*

# Transport Packages

- **MT3DMS:** Three-dimensional transport model for simulating advection, dispersion, and chemical reactions of dissolved constituents.
- **MODFLOW-SURFACT:** Enhanced simulations of complex saturated/unsaturated subsurface flow and transport processes.
- **SEAWAT:** Three-dimensional variable-density groundwater flow coupled with multi-species solute and heat transport.
- **RT3D:** Reactive transport simulations.
- **MT3D99\*:** An enhanced version of MT3DMS, that includes support for implicit solver, TVD Solution scheme, dual-porosity advection-diffusion, Non-equilibrium Sorption and Monod Kinetics, and Multi-species Reactions, including First-Order Parent-Daughter chain reactions, and Instantaneous Reactions among species.
- **PHT3D\*:** A multi-component transport model for three-dimensional reactive transport in saturated porous media.

\* *Available with the Classic interface.*

# Parameter Estimation and Sensitivity Analysis

**PEST v12.3:** Automated parameter estimation, calibration and sensitivity analysis

Defining PEST inputs & interpreting results is easy with an intuitive, easy-to-use interface that guides you through the PEST process from start to finish.

# Data Visualization

## Designed with 3D Visualization Technology:

- Visualize all data, including conceptual objects, numerical grids, input data (property zones, boundary condition cells) and output data (calculated heads, pathlines, water table) in 2D, 3D & multi-view (FlexViewer) displays.
- Create cut-away & cross-sectional views that allow you to see inside of the model.
- Generate 3D animations & movies for reports or for web publishing.
- Add isolines, contours, pathlines & color shading to viewers.
- Drape raster images (site maps, satellite imagery) over 3D surfaces to show relief.

# Uncertainty & Multiple Scenarios

**Visual MODFLOW Flex allows to assess uncertainty & comparisons of multiple modeling scenarios – all within a single project.**

- Manage multiple model scenarios in a single project.
- Easily generate multiple models in parallel for evaluating alternative hydrogeologic interpretations and hypotheses.
- Direct visual and numerical comparisons between different modeling scenarios.
- Calculate head differences between multiple model runs, with the same or different grid sizes.
- Compare & analyze multiple modeling scenarios for selecting the best, and most realistic model.

# Flexible Grid Options

**Visual MODFLOW Flex provides different grid types and choose the best, & most stable model.**

- Choose various structured and unstructured grid types to accommodate a wide range of applications and geologic conditions (e.g., pinch outs, discontinuities).
- Improve simulation accuracy, reduce runtimes and increase model stability with unstructured grid types (MODFLOW-USG).
- Generate multiple grids within the same project and compare side-by-side in 2D or 3D.
- Perform grid refinement around areas of interest.
- Create faster & more stable models with nested child grids that contain local horizontal & vertical refinement (MODFLOW-LGR).

# GIS Integration

- Easy to construct grid-independent hydrogeologic conceptual model using existing GIS datasets.
- Quick & easy data import facility from all common file types and formats.
- Automatic conversion of coordinate system & units on import.
- Automatic data validation & error checking on import.
- Option to define model boundaries, property zones, boundary conditions & attributes from imported GIS data.

# **Integrated 3D Conceptual & Numerical GW Modeling**

**Visual MODFLOW Flex provides a seamless transition from raw data through conceptualization model to numerical model in a single modeling environment:**

- Define complex geology & model layers using borehole log data and cross-sections.
- Interpret GIS data to define hydrogeologic properties & boundaries independent of the grid.
- Run, analyze, & validate model results with raw data in 2D, 3D, & cross-sectional views.
- Create multiple conceptualizations; generate numerous grids & model scenarios in parallel all in a single project.
- Assess uncertainty though comparing heads from multiple model runs in 2D, 3D or chart views.



# Visual MODFLOW Flex: Feature Comparison

Features	Basic	Professional	Premium
MODFLOW-2000, 2005, NWT	√	√	√
MODFLOW-USG (Un-Structured Grids)		√	√
ZoneBudget	√	√	√
MODPATH	√	√	√
MOD-PATH3DU		√	√
Enhanced 3D Visualization	√	√	√
MT3DMS	√	√	√
RT3D	√	√	√
Local Grid Refinement (MODFLOW-LGR)		√	√
Conceptual Modeling		√	√
Multiple Grid Types		√	√
Compare results from multiple model runs		√	√
Modeling Scenarios in a Project	One	Unlimited	Unlimited
Build and run bigger models (Native 64-bit Support)		√	√
3D Animation and Movie Generation		√	√
PEST		√	√
SEAWAT		√	√
SAMG Solver (Serial)		√	
SAMG Solver (Parallel)			√
FEFLOW Model Generation (.FEM file)			√
MODFLOW-SURFACT	Add-On	Add-On	Add-On

# MODFLOW Packages

Functionality	Package & Processes Name	Purpose
<b>BAS</b>	Basic Package	<p>These include</p> <ul style="list-style-type: none"> <li>• model domain (X,Y,Z extents), no. of layers, etc.</li> <li>• locations of active, inactive, and specified head cells</li> <li>• head stored in inactive cells, and</li> <li>• initial heads in all cells.</li> <li>• Includes boundary locations, time-step length, initial conditions, and printing of results.</li> </ul>
<b>Flow component packages: account all groundwater flow components.</b>		
<b>BCF</b>	Block-Centered Flow Package	Specify properties controlling flow between cells.
<b>LPF</b>	Layer-Property Flow Package	Specify properties of porous media of cells
<b>HFB</b>	Horizontal Flow Barrier Package	Simulate barriers to flow such as slurry walls by reducing the conductance between individual pairs of cells.
<b>CHD</b>	Time-Variant Specified-Head Option	Simulate specified head boundaries that can change within or between stress periods.
<b>RIV</b>	River Package	Simulate head-dependent flux boundaries.
<b>DRN</b>	Drain Package	Simulate head-dependent flux boundaries.
<b>WEL</b>	Well Package	Simulate a specified flux to individual cells and specified in units of length <sup>3</sup> /time.
<b>GHB</b>	General Head Boundary Package	Simulate head-dependent flux boundaries.
<b>RCH</b>	Recharge Package	Simulate a specified flux distributed over the top of the model and specified in units of length/time.
<b>EVT</b>	Evapotranspiration Package	Simulate a head-dependent flux out of the model distributed over the top of the model and specified in units of length/time.

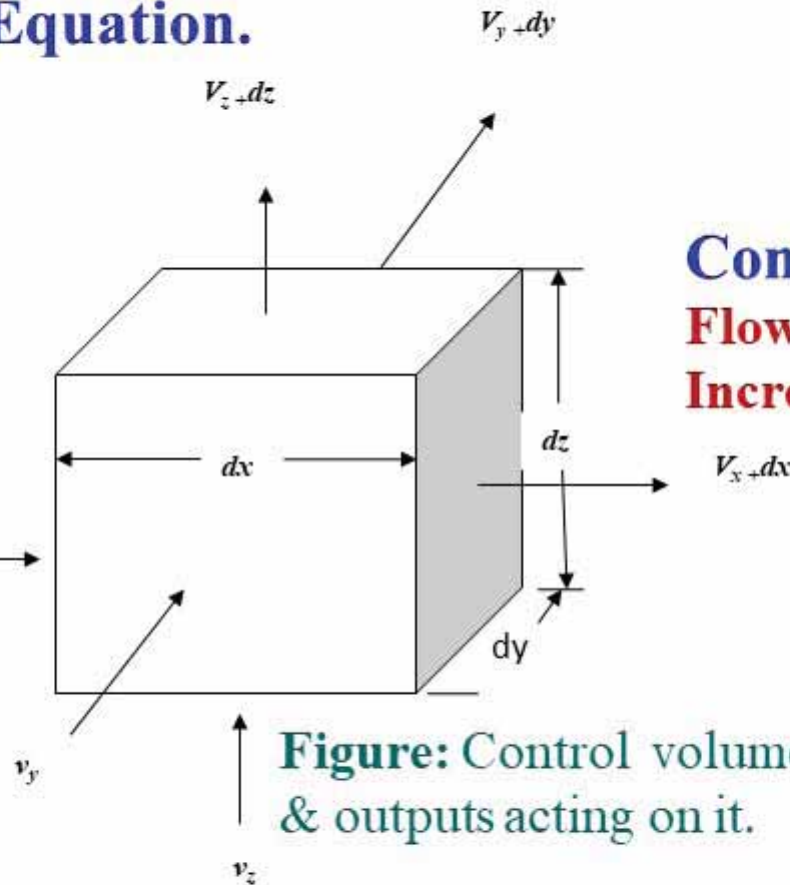
# Numerical Schemes

- Groundwater equation follows: Integration of **Darcy's Equation** into **Continuity Equation**.

**Darcy's equation:**

$$V_x = K_{xx} \frac{dh}{dx}$$

$v_x$  →



**Continuity equation:**  
**Flow in – Flow out = Increase in storage**

**Figure:** Control volume showing inputs & outputs acting on it.

$$\frac{\partial}{\partial x} \left( K_x \frac{\partial h}{\partial x} \right) + \frac{\partial}{\partial y} \left( K_y \frac{\partial h}{\partial y} \right) + \frac{\partial}{\partial z} \left( K_z \frac{\partial h}{\partial z} \right) = S_s \frac{\partial h}{\partial t}$$

## Finite Difference Scheme in MODFLOW

$$\begin{aligned}
 & CR_{i,j-\frac{1}{2},k} (h_{i,j-1,k}^m - h_{i,j,k}^m) + CR_{i,j+\frac{1}{2},k} (h_{i,j+1,k}^m - h_{i,j,k}^m) + \\
 & CC_{i-\frac{1}{2},j,k} (h_{i-1,j,k}^m - h_{i,j,k}^m) + CC_{i+\frac{1}{2},j,k} (h_{i+1,j,k}^m - h_{i,j,k}^m) + \\
 & CV_{i,j,k-\frac{1}{2}} (h_{i,j,k-1}^m - h_{i,j,k}^m) + CV_{i,j,k+\frac{1}{2}} (h_{i,j,k+1}^m - h_{i,j,k}^m) + \\
 & P_{i,j,k} h_{i,j,k}^m + Q_{i,j,k} = SS_{i,j,k} (\Delta r_j \Delta c_i \Delta v_k) \frac{h_{i,j,k}^m - h_{i,j,k}^{m-1}}{t^m - t^{m-1}}
 \end{aligned}$$

- $h_{i,j,k}^m$  is the hydraulic head at cell  $i,j,k$  at time step  $m$ ;
- $CV$ ,  $CR$  and  $CC$  are the hydraulic or branch conductances between node  $i,j,k$  and a neighbouring node;
- $P_{i,j,k}$  is the sum of coefficients of head from source and sink terms;
- $Q_{i,j,k}$  is the sum of constants from source and sink terms; for flow out of the groundwater system (such as pumping) and flow in (such as injection);
- $SS_{i,j,k}$  is the specific storage;
- $\Delta c_i \Delta r_j$  and  $\Delta v_k$  are the dimensions of cell  $i,j,k$ , which, when multiplied, represent the volume of the cell; and
- $t^m$  is the time at time step  $m$ .

## Finite Difference Scheme in MODFLOW

$$\begin{aligned}
 & CV_{i,j,k-\frac{1}{2}} h_{i,j,k-1}^m + CC_{i-\frac{1}{2},j,k} h_{i-1,j,k}^m + CR_{i,j-\frac{1}{2},k} h_{i,j-1,k}^m \\
 & + \left( -CV_{i,j,k-\frac{1}{2}} - CC_{i-\frac{1}{2},j,k} - CR_{i,j-\frac{1}{2},k} - CR_{i,j+\frac{1}{2},k} - CC_{i+\frac{1}{2},j,k} - CV_{i,j,k+\frac{1}{2}} + HCOF_{i,j,k} \right) h_{i,j,k}^m \\
 & + CR_{i,j+\frac{1}{2},k} h_{i,j+1,k}^m + CC_{i+\frac{1}{2},j,k} h_{i+1,j,k}^m + CV_{i,j,k+\frac{1}{2}} h_{i,j,k+1}^m = RHS_{i,j,k}
 \end{aligned}$$

Where,

$$HCOF_{i,j,k} = P_{i,j,k} - \frac{SS_{i,j,k} \Delta r_j \Delta c_i \Delta k}{t^m - t^{m-1}}$$

$$RHS_{i,j,k} = -Q_{i,j,k} - SS_{i,j,k} \Delta r_j \Delta c_i \Delta v_k \frac{h_{i,j,k}^{m-1}}{t^m - t^{m-1}}$$

**In matrix form:  $\mathbf{A} \mathbf{h} = \mathbf{q}$**

Where,

**A** is a matrix of the coefficients of head for all active nodes.

**h** is a vector of head values at the end of time step **m** for all nodes.

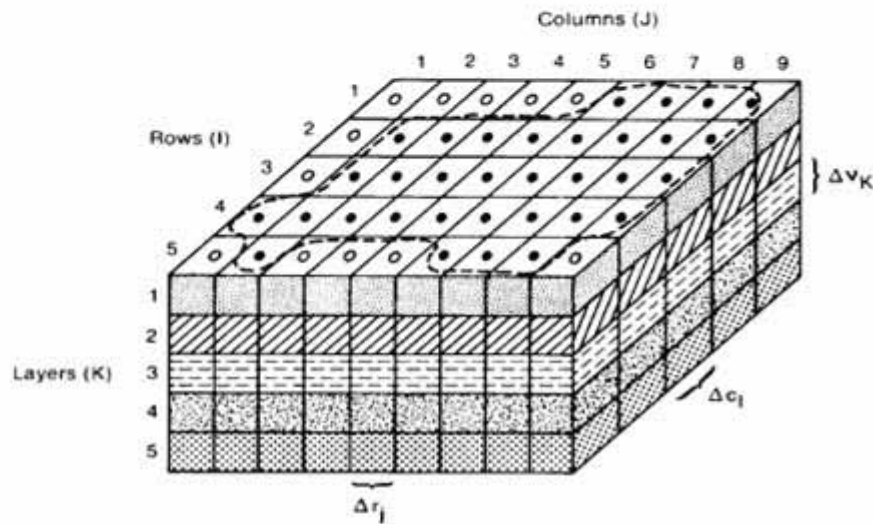
**q** is a vector of the constant terms, *RHS*, for all nodes of the grid.

# MODFLOW Solvers

Functionality	Package Name	Purpose
<b>Solver Packages</b>		
<b>SIP</b>	Strongly Implicit Procedure	Iteratively solves the system of finite-difference equations.
<b>PCG</b>	Preconditioned-Conjugate Gradient	Solve the finite difference equations in each step of a MODFLOW stress period.
<b>DE4</b>	Direct Solver	Solve the finite difference equations in each step of a MODFLOW stress period.
<b>GMG</b>	Geometric Multi-Grid	Solve the finite difference equations in each step of a MODFLOW stress period.
<b>Other Flow Packages</b>		
<b>STR</b>	Stream	Simulate streams in a model.
<b>FHB</b>	Flow and Head Boundary	Used for specified head cells and specified flow cells, whose properties can vary within a stress period
<b>IBS</b>	Inter-Bed Storage	Simulate compaction of low-permeability interbeds within layers.
<b>HUF</b>	Hydrogeologic-Unit Flow	Used to specify properties controlling flow between cells.
<b>RES</b>	Reservoir	Simulates leakage between a reservoir and an underlying groundwater system .
<b>OBS</b>	Observation	Compare model-generated values of head, flux, or advective transport with observed values.
<b>SFR</b>	Stream Flow-Routing	Simulate streams in a model. Unsaturated flow beneath streams can also be simulated.
<b>LAK</b>	Lake	Used to simulate lakes.
<b>UZF</b>	Unsaturated Zone	Simulates vertical flow of water through unsaturated zone to the saturated zone.
<b>SWI2</b>	Seawater Intrusion	Allows three-dimensional vertically integrated variable-density groundwater flow and seawater intrusion in coastal multi-aquifer systems.

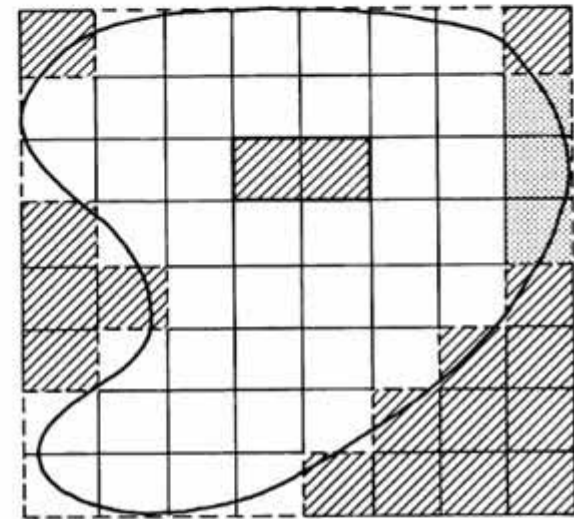
# Schematization in MODFLOW

3-Dimensional grids considered in schematization. Discretized aquifer showing boundaries and cell designations.



Explanation

- Aquifer Boundary
- Active Cell
- Inactive Cell
- $\Delta r_j$  Dimension of Cell Along the Row Direction. Subscript (J) Indicates the Number of the
- $\Delta c_l$  Dimension of Cell Along the Column Direction. Subscript (I) Indicates the Number of
- $\Delta v_k$  Dimension of the Cell Along the Vertical Direction. Subscript (K) Indicates the Numbe



Explanation

- Aquifer Boundary
- Model Impermeable Boundary
- ▨ Inactive Cell
- ▤ Constant-Head Cell
- Variable-Head Cell

# Modflex Overview

The screenshot displays the Visual MODFLOW Flex software interface. The main window is titled "Visual MODFLOW Flex - [example] - [DRUMCO grid-Run]". The interface is divided into several panels:

- Data Objects:** A tree view on the left showing a folder named "Data" containing six "dumco Elevation" objects (a1 through a6) and "VMod Imported Wells".
- Workflow Navigation:** A central panel with navigation icons (back, forward, home) and a list of workflow steps including "Define Modeling Objectives", "Define Numerical Model", "Create Grid", "Input Model", "View/Edit Grid", "Define Properties", "Define Boundary Conditions", "Select the Next Step", "Translate", "Run Numerical Engine", and "View Results".
- Model Explorer:** A tree view at the bottom left showing the project structure, including "Wells", "DRUMCO grid", "Run", "Inputs", "Properties", "Flow", "Boundary Conditions", and "Rivers".
- Inspect/Define Properties:** A panel on the right with settings for "Views" (Layer, Flow, Column, 3D), "Toolbox" (Conductivity, Zone), and "Database".
- Viewers:** Two visualization windows: "Column View" showing a 3D bar chart of data across columns, and "Layer View" showing a 2D grid with colored zones (blue, green, red).

Callouts with yellow boxes point to these key features: "Data Objects", "Workflow Navigation", "Workflow Steps", "Model Explorer", and "Viewers".

— Data Objects



## Features

VMOD Flex supports the following coordinate systems:

- Geographic coordinate systems (data import only)
- Projected coordinate systems: UTM, StatePlane
- Local Cartesian

## **Data Import Formats**

Import spatial and attribute data from a wide variety of data types including:

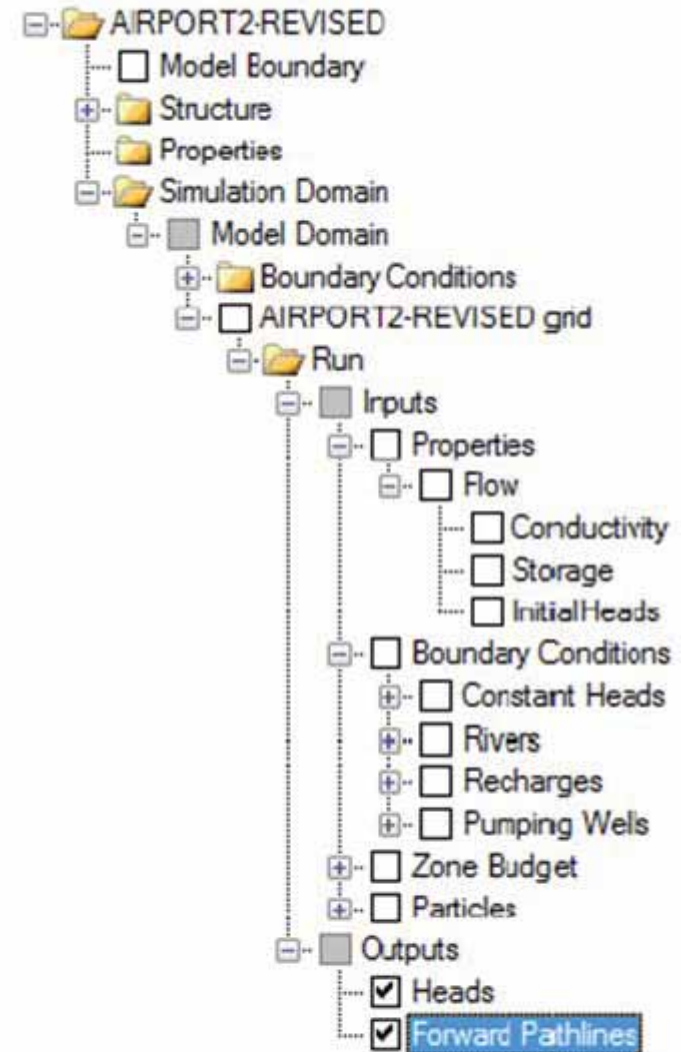
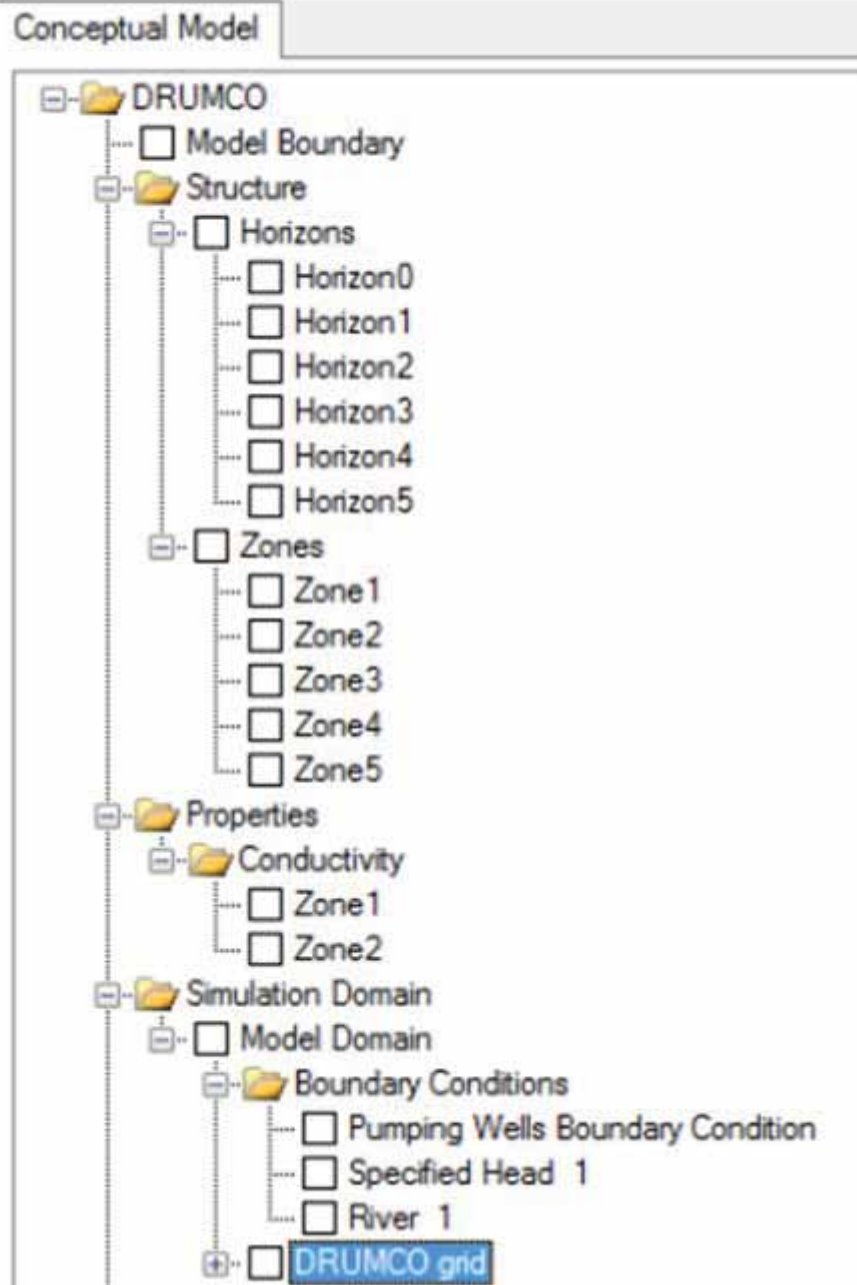
- Points (.XLS, .TXT, .CSV, .MDB, .SHP, .DXF, .TRP)
- Polygons (.SHP, .DXF)
- Polylines (.SHP, .DXF)
- 3D Gridded Data (.HDS, .DAT)
- Raster Images (.BMP, .TIF, .JPG)
- Time Schedules (.XLS)
- Surfaces (.DEM, .GRD, .TXT, .ASC)
- Hydro GeoAnalyst (HGA) Cross Sections (.3XS)
- Vertical and Horizontal Wells (.XLS )

## **Boundary Conditions in Flex**

Support for the following boundary conditions:

- Pumping Wells
- Specified Head
- River
- General Head
- Drain
- Recharge
- Evapotranspiration
- Lake
- Specified Flux
- Streams

# Model Tree for Conceptual & Numerical Model

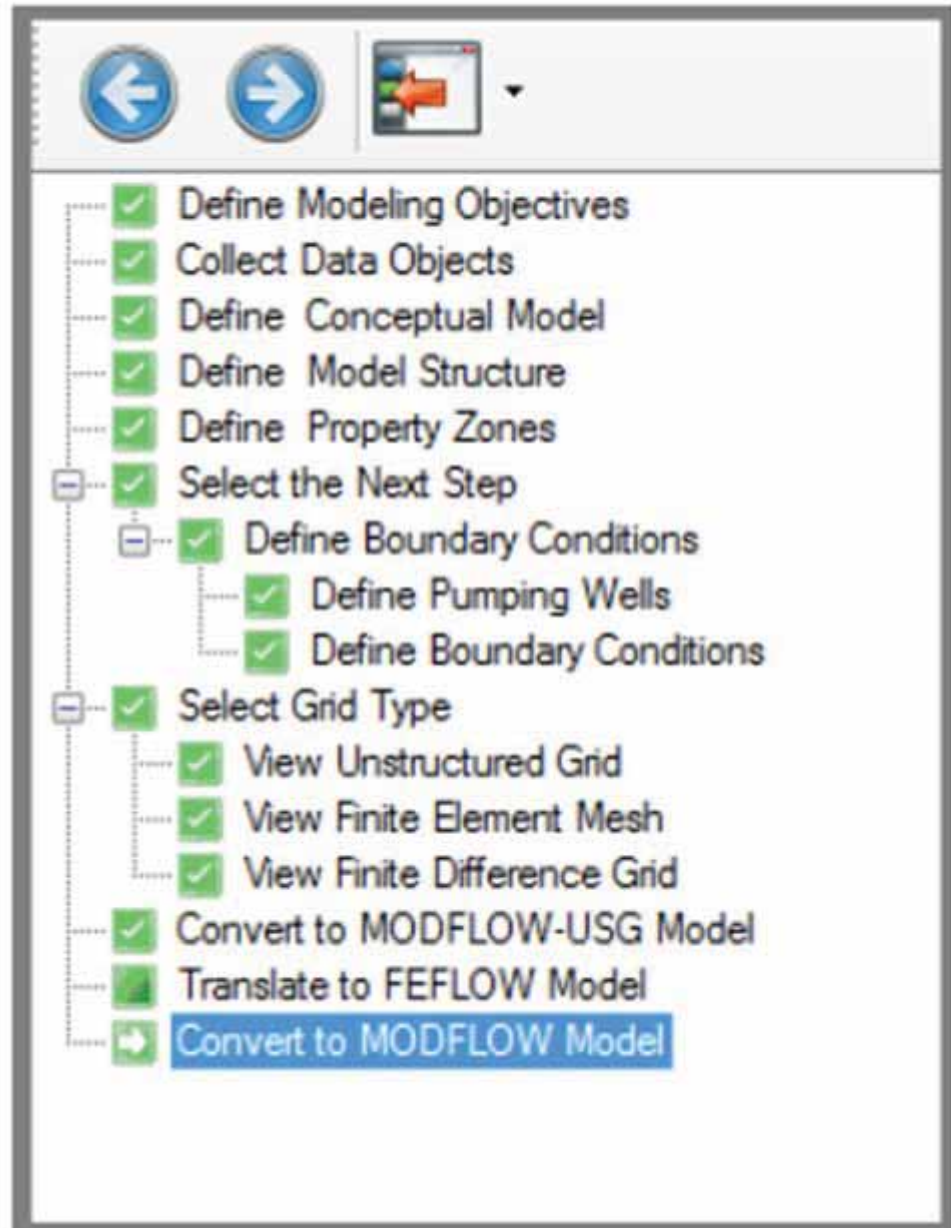


## **Flex Viewer**

The "Flex" Viewer: which is available in the numerical modeling workflow, and consists of a combination of a Layer, Row and Column view and 3D View; the individual views can be shown/hidden.

# Workflows

**GW Modeling consists of a series of steps that must be completed in a particular sequence.**



## Navigation of Workflow



Go to the Previous Step in the workflow



Go to the Next Step in the workflow



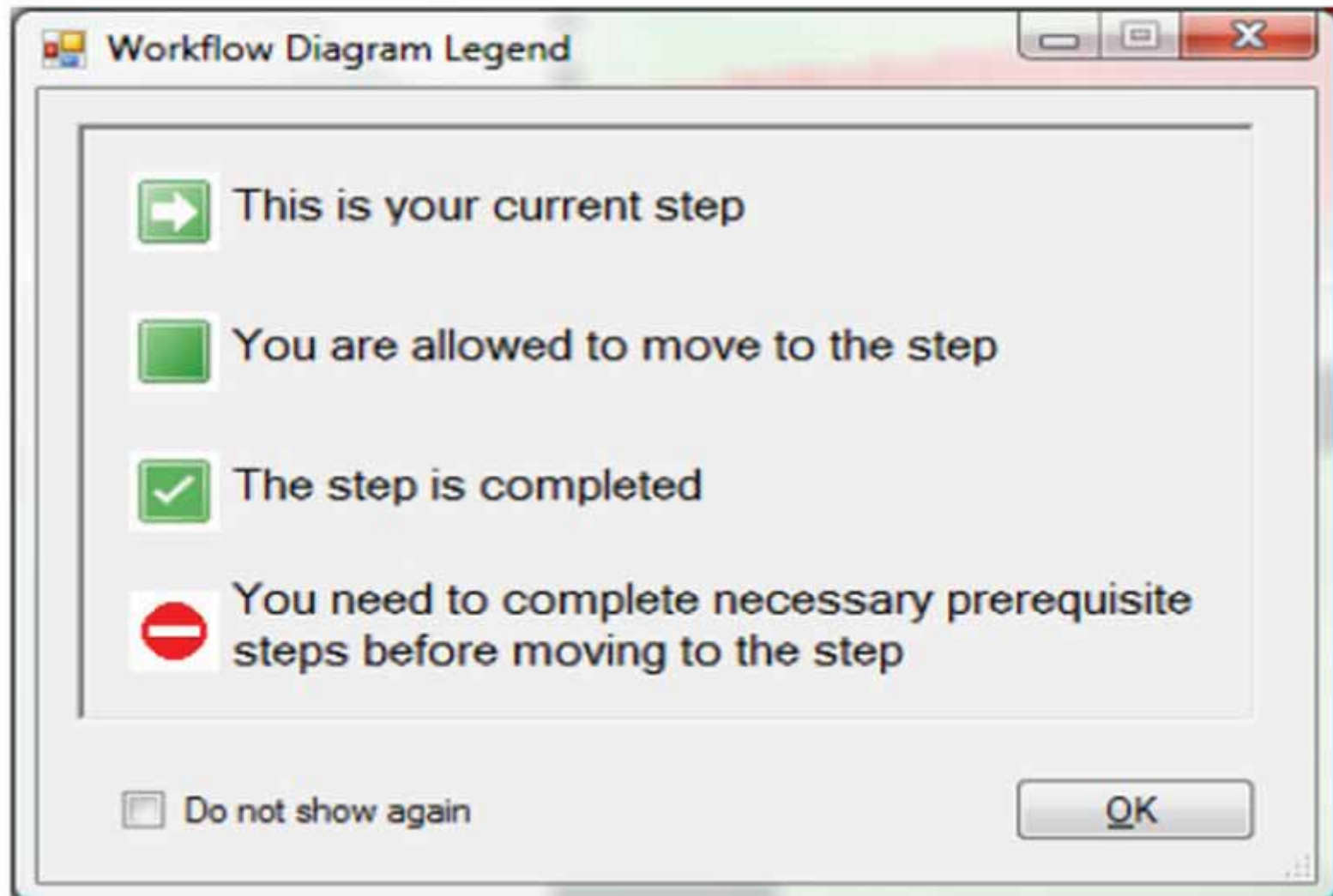
Hide the workflow panel; ideal when you want to maximize the viewing space for visualization and editing.



Expand the workflow panel; this button is only available when the workflow panel has been collapsed.

## 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.





# Project Creation

Create Project

Project Information

Name: \*

Exercise

Data Repository: \*

D:\Documents\VMODFlex

Description:

Project Coordinate

Coordinate Systems: \*

Local Cartesian

Datum: \*

World Geodetic System 1984

Units \*

Unit Settings

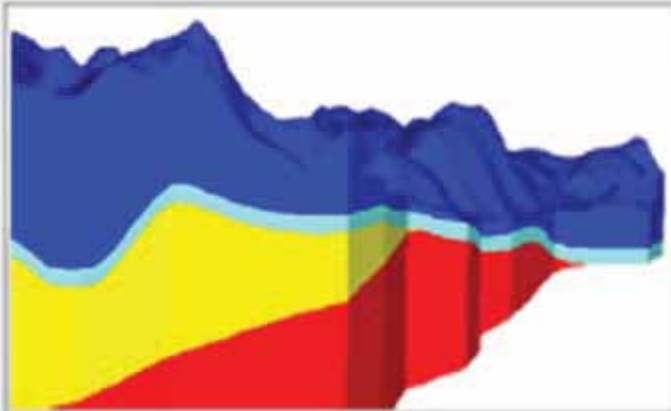
Conductivity	m/s
Length	m
Pumping Rate	$m^3/d$
Recharge	mm/yr
Specific Storage	1/m
Time	day

Conductivity

OK Cancel Help

# Type of Model

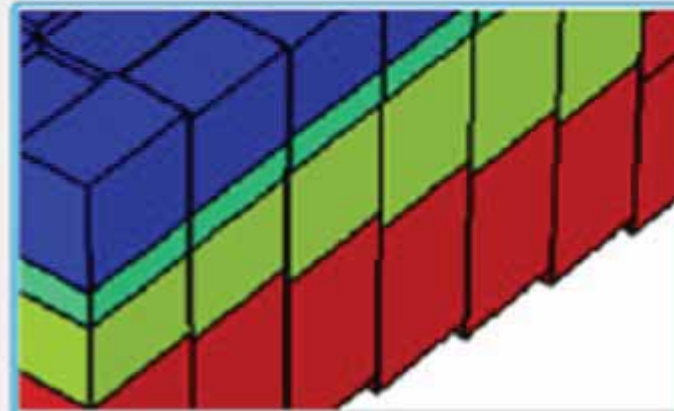
## Select Modeling Scenario



Conceptual Modeling

- Import raw GIS data and interpret in 2D/3D
- Build geological models and flow boundaries
- Design structured and unstructured grids
- Build inputs for Local Grid Refinement (LGR)

Recommended for new groundwater models



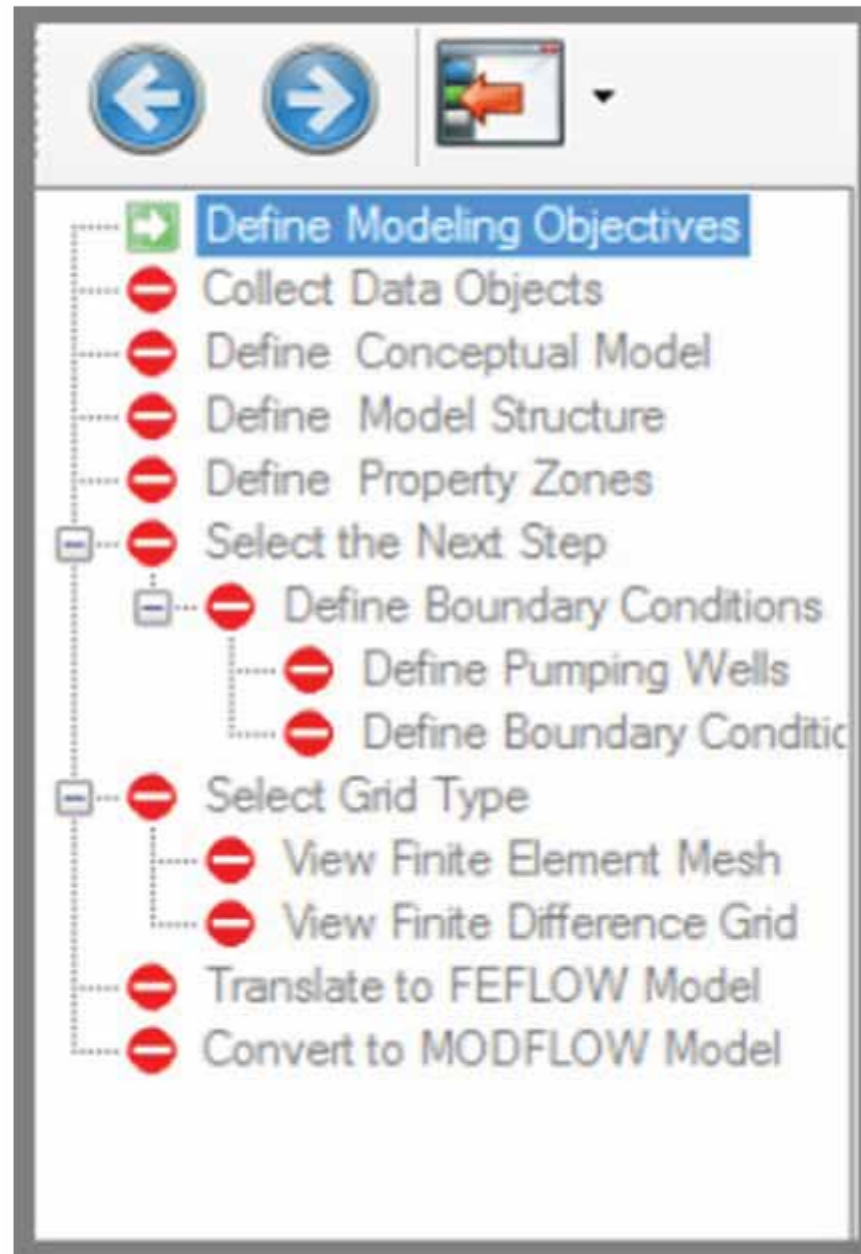
Numerical Modeling

- Import Visual MODFLOW projects
- Import USGS MODFLOW data sets

Recommended for existing flow models.

NOTE: If you need to maintain a model that uses PEST, Transport, or SEAWAT, you must continue to use VMOD Classic interface.

# Steps to Follow



# Define Objective of Model

Visual MODFLOW File - [Items]

File Tools Workflow Window Help

Conceptual Model

Define Modeling Objectives

Review:  Transport Active

Simulation type: Groundwater Flow

Available Engines:

- USGS MODFLOW 2000 from SMS
- USGS MODFLOW 2005 from SMS
- USGS MODFLOW LGR from SMS
- ZONEDUCGET
- MODPATH

Start Date: 5/15/2013

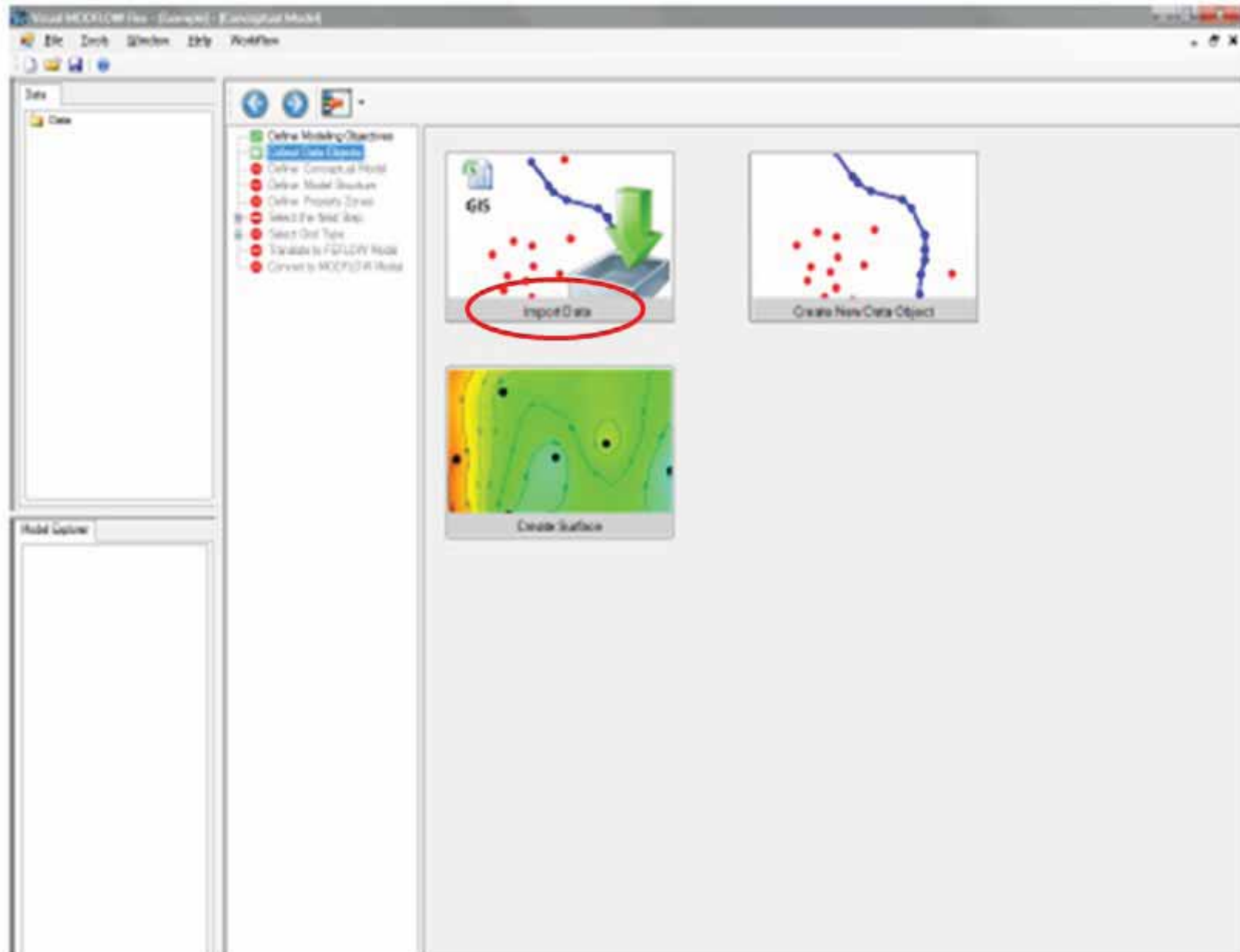
Property Settings:

- Conductivity
  - $K_x$  (m/s): 0.0001
  - $K_y$  (m/s): 0.0001
  - $K_z$  (m/s): 1E-05
- Storage
  - $S_h$  (T/N): 1E-05
  - $S_y$ : 0.2

Species Parameters (Model Parameters)

Name	Description	Mobile	SOCM (mg/L)
Comp001	Comp001	<input checked="" type="checkbox"/>	0

# All Types of Data Import

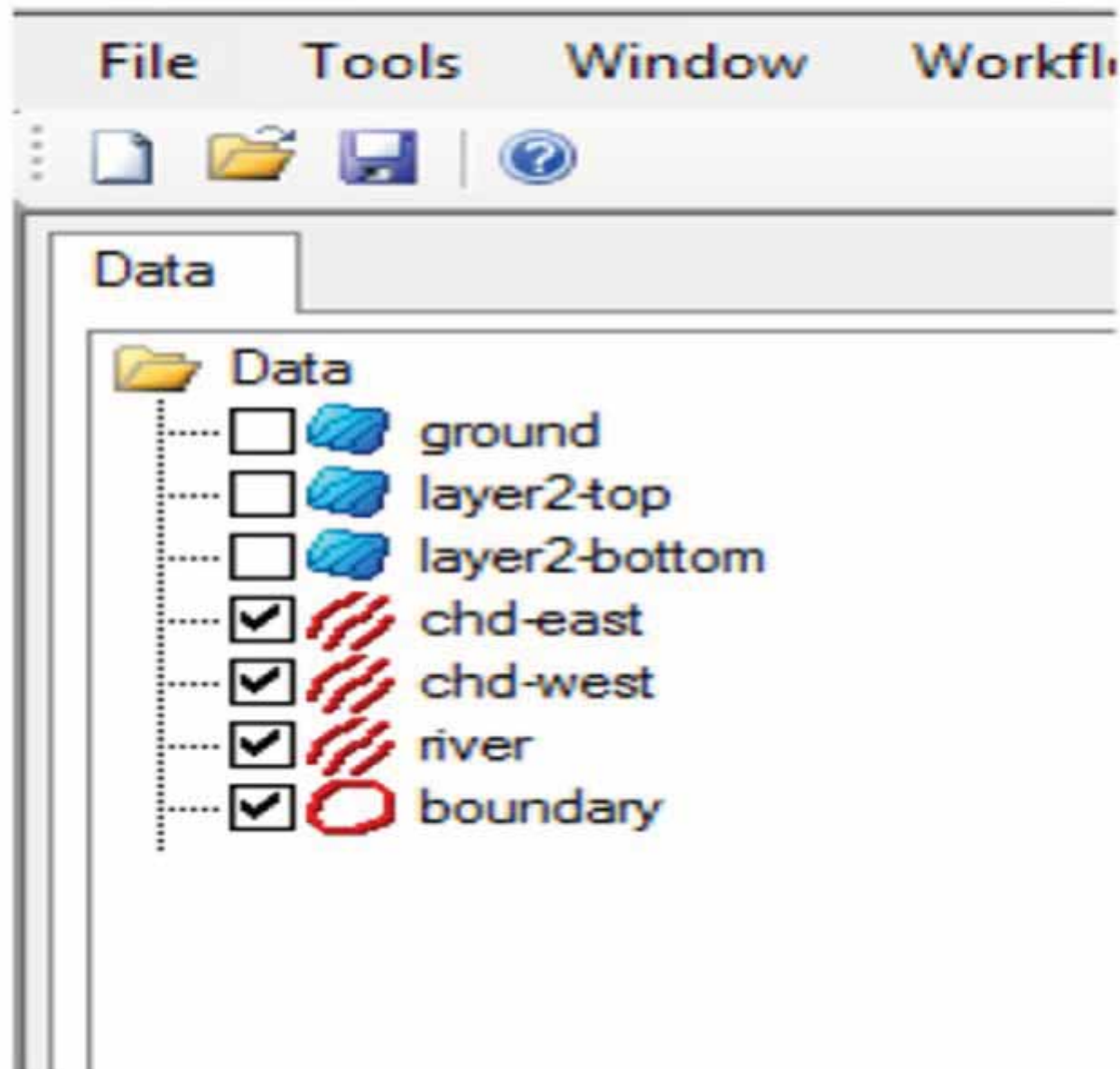


# Data Type Selection

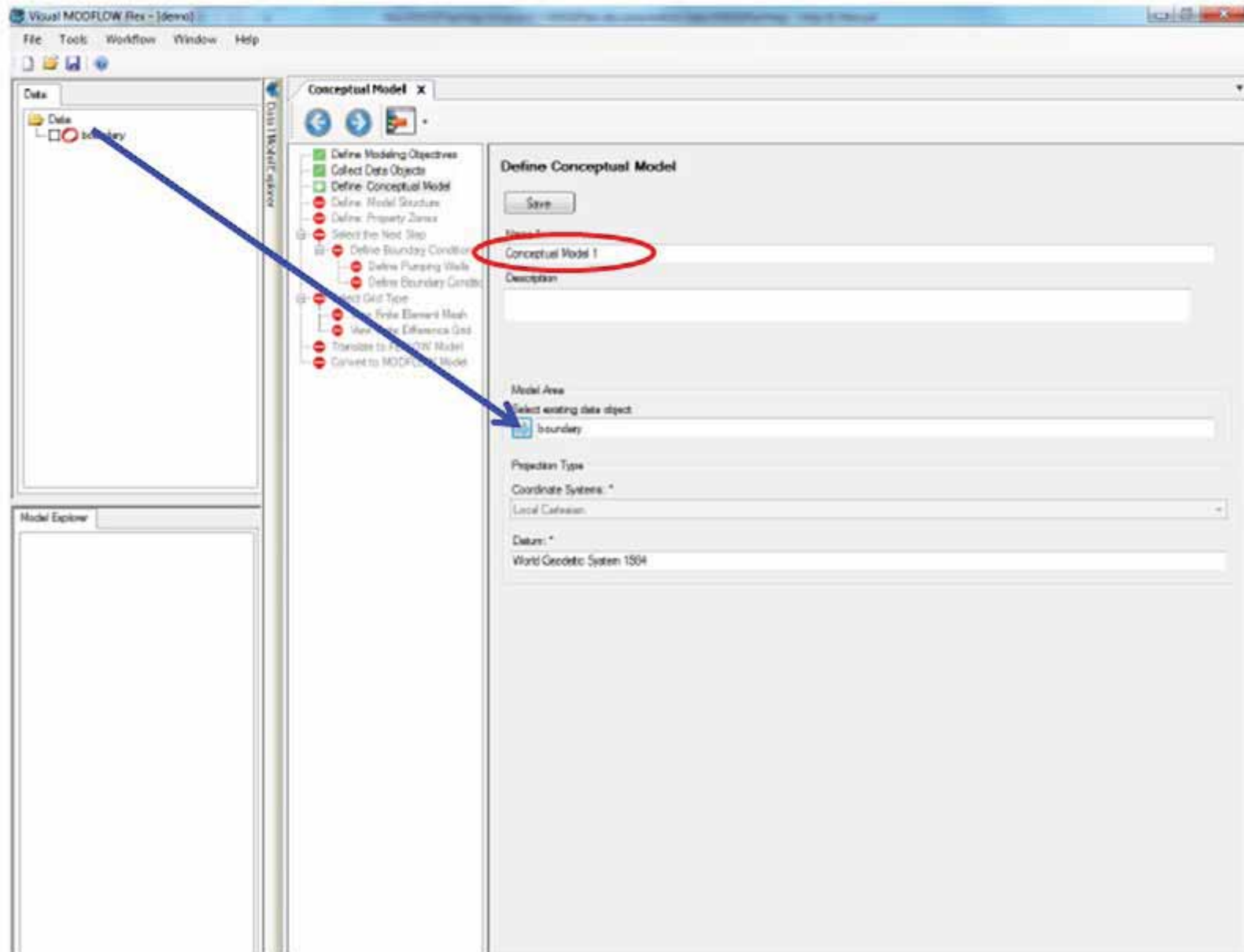
The image shows a 'Data Import' dialog box with the following fields and controls:

- Data Source:** A section header.
- Data Type:** A dropdown menu with 'Point' selected.
- Source File:** A text input field with a browse button ('...').
- Name:** A text input field.
- Description:** A large text area.
- Buttons:** '< Back', 'Next >', 'Cancel', and 'Help'.

# Imported Data



# Add Imported Data to the Conceptual Model





# Define Structure

**Define Conceptual Model Structure**

Horizon Information

Surface	Name	Type
ground	Horizon1	Erosional
layer2-top	Horizon2	Conformable
layer2-bottom	Horizon1	Conformable

# Preview of Imported Data

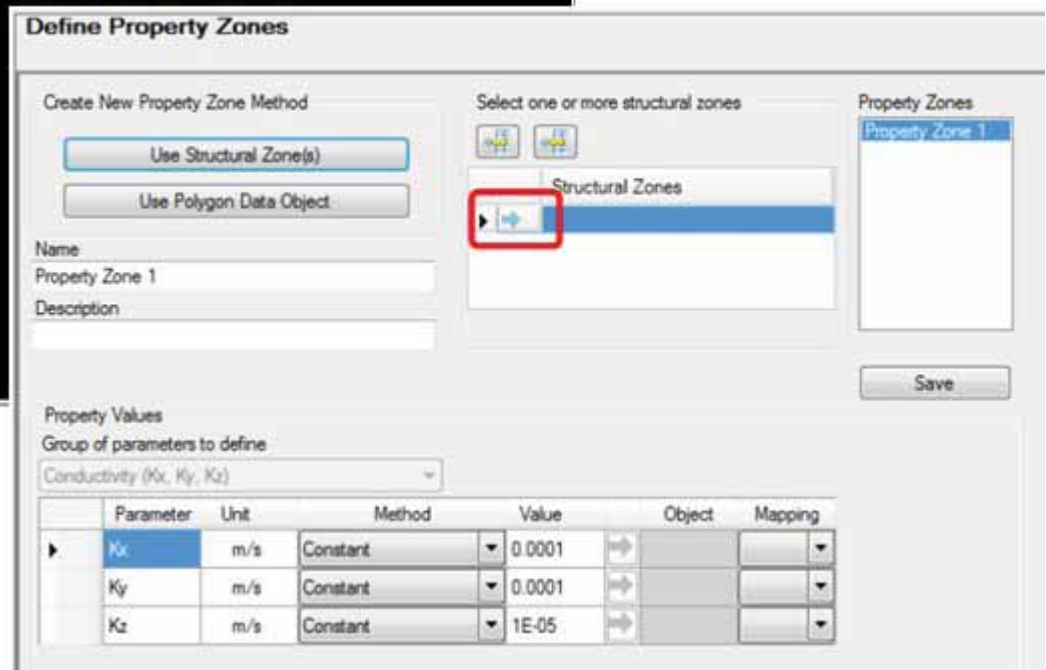
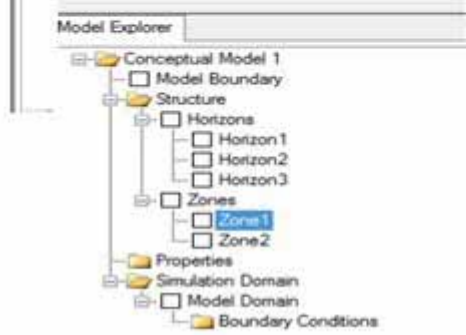
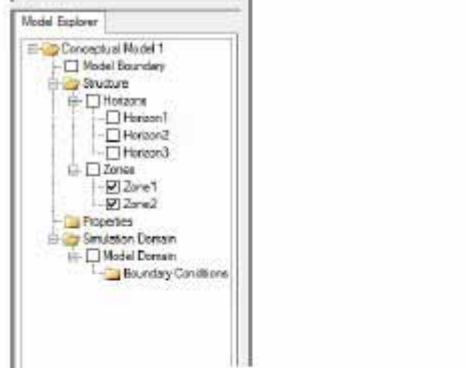
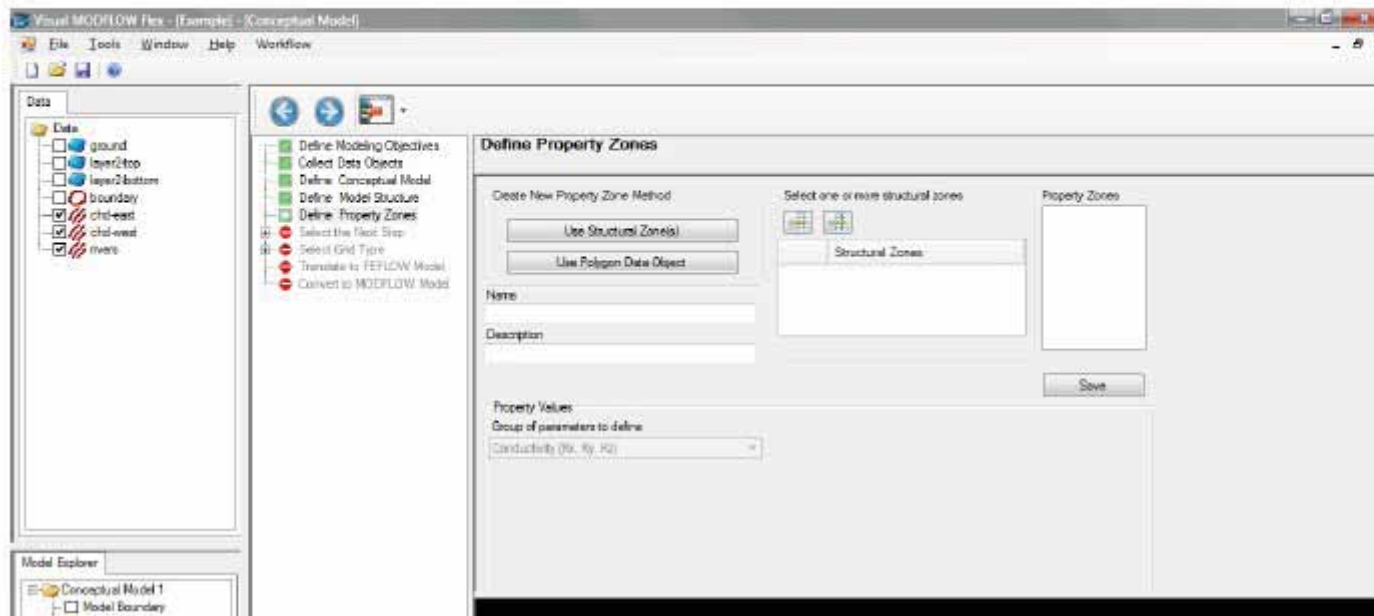
The screenshot displays a software interface for defining a conceptual model structure. On the left, a vertical list of steps is shown, with the first three steps completed: 'Define Modeling Objectives', 'Collect Data Objects', and 'Define Conceptual Model'. The current step is 'Define Model Structure'. Below this list are several options for defining property zones, selecting the next step, selecting grid type, translating to FEFLOW model, and converting to MODFLOW model.

The main area is titled 'Define Conceptual Model Structure' and contains a 'Preview' button and a 'Create' button. Below these buttons is a table labeled 'Horizon Information' with the following data:

Surface	Name	Type
ground	Horizon1	Erosional
layer2-top	Horizon2	Conformable
layer2-bottom	Horizon3	Conformable

Below the table is a 'Preview' section with a 'Magnification' slider set to 1. The preview area shows a 2D plot of imported data, which is a color-coded map of a region. The x-axis is labeled 'X' and ranges from 0 to 15000, with major ticks at 5000, 10000, and 15000. The y-axis is labeled 'Y' and ranges from 0 to 15000, with major ticks at 5000, 10000, and 15000. The plot shows a blue river channel flowing from the top left towards the bottom center, and a red/orange area on the right side. A red arrow points from the 'Create' button to the plot.

# Prepare Property Zones & Assign Data



# Assign Boundaries

The screenshot displays the Visual MODFLOW Flex software interface. The main window title is "Visual MODFLOW Flex - [Example] - [Conceptual Model]". The menu bar includes "File", "Tools", "Window", "Help", and "Workflow".

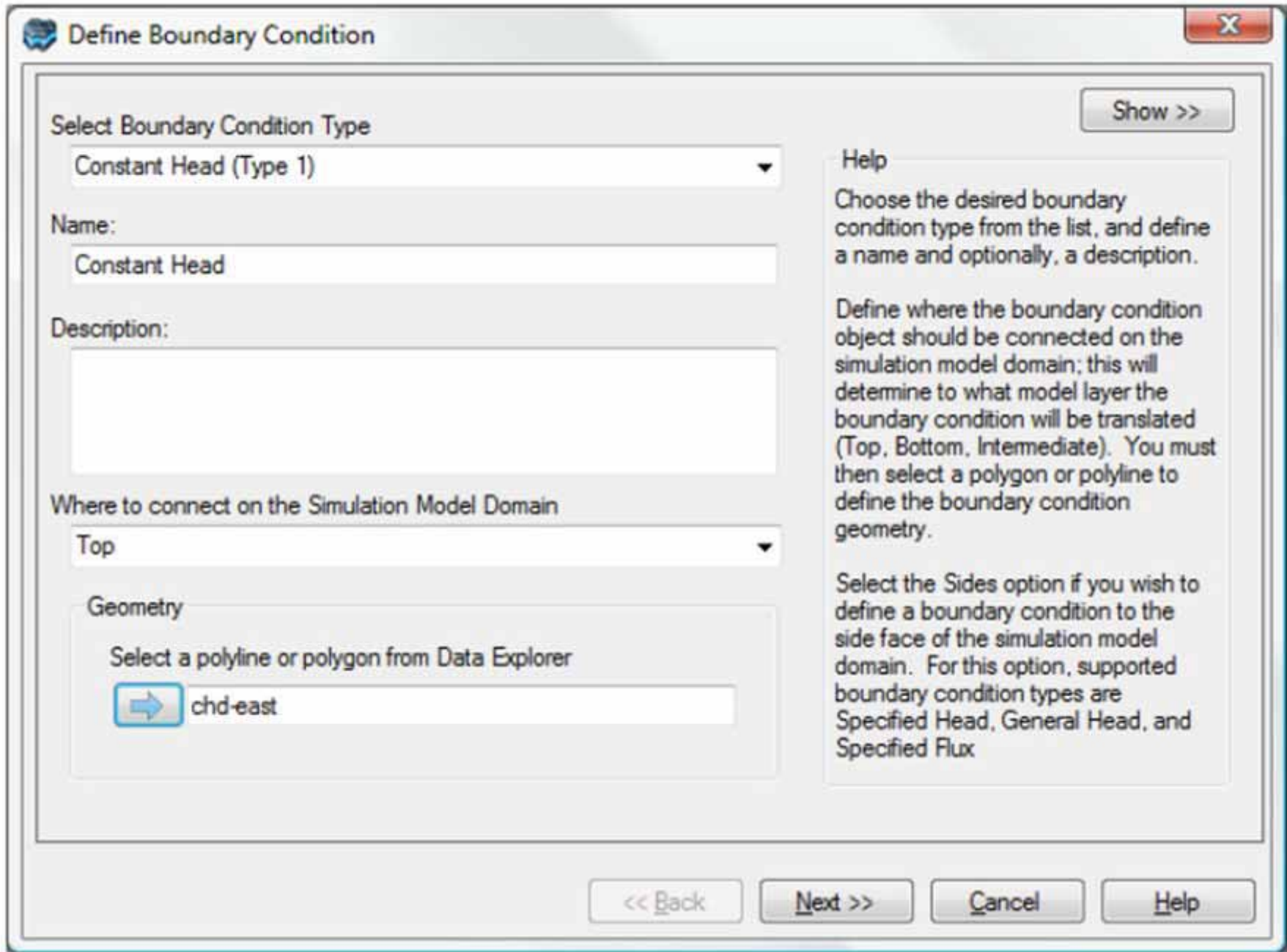
**Data Panel (Left):** A tree view showing data objects: ground, layer2top, layer2bottom, boundary, chd east, chd west, and rivak. The "boundary" object is selected.

**Model Explorer (Bottom Left):** A tree view showing the model structure: Conceptual Model 1, Model Boundary, Structure (Horizon 1, 2, 3), Zones (Zone 1, 2), Properties (Conductivity: Property Zone 1, 2), Simulation Domain, Model Domain, and Boundary Conditions.

**Workflow Panel (Top Center):** A list of steps in the workflow. The step "Assign the Grid Size" is highlighted in blue. Other steps include: Define Modeling Objectives, Collect Data Objects, Define Conceptual Model, Define Model Structure, Define Property Zones, Define Boundary Conditions, Select Grid Type, Transfer to FEFLOW Model, and Convert to MODFLOW Model.

**Main Canvas (Right):** The main workspace shows a map titled "Select the Next Step". On the left, a legend lists boundary types: CHD, DRN, GHB, and RCH. The map displays a yellow and red area with a red boundary line. A red circle highlights the "Define Boundary Conditions" button at the bottom of the map. To the right, a "Select Grid Type" panel shows two grid options: a regular rectangular grid and an irregular triangular grid.

# Constant Head Boundary

A software dialog box titled "Define Boundary Condition" with a close button (X) in the top right corner. The dialog is divided into several sections. On the left, there is a "Select Boundary Condition Type" dropdown menu set to "Constant Head (Type 1)". Below it is a "Name:" text field containing "Constant Head". Underneath is a "Description:" text area which is currently empty. Further down is a "Where to connect on the Simulation Model Domain" dropdown menu set to "Top". At the bottom left is a "Geometry" section with the instruction "Select a polyline or polygon from Data Explorer" and a text field containing "chd-east" with a blue arrow icon to its left. On the right side of the dialog is a "Help" section with a "Show >>" button above it. The help text contains three paragraphs: the first explains how to choose a boundary condition type and name; the second explains how to define the connection location (Top, Bottom, Intermediate) and geometry; the third lists supported boundary condition types for the "Sides" option: Specified Head, General Head, and Specified Flux. At the bottom of the dialog are four buttons: "<< Back", "Next >>", "Cancel", and "Help".

Define Boundary Condition

Select Boundary Condition Type

Constant Head (Type 1)

Name:

Constant Head

Description:

Where to connect on the Simulation Model Domain

Top

Geometry

Select a polyline or polygon from Data Explorer

chd-east

Show >>

Help

Choose the desired boundary condition type from the list, and define a name and optionally, a description.

Define where the boundary condition object should be connected on the simulation model domain; this will determine to what model layer the boundary condition will be translated (Top, Bottom, Intermediate). You must then select a polygon or polyline to define the boundary condition geometry.

Select the Sides option if you wish to define a boundary condition to the side face of the simulation model domain. For this option, supported boundary condition types are Specified Head, General Head, and Specified Flux

<< Back   Next >>   Cancel   Help

# Edit Boundary Condition

Edit Boundary Condition

Polylines: PLine0

Zones: Polyline0.Zone0

Points: Point1

Show >>

Select how the attributes are defined

- Define for the entire zone.
- Define values at vertices (Linear Interpolation)
  - Just start and end points.
  - All vertices.

Starting Head	Ending Head
Static	
Constant	
347	347

From 3D gridded   From shapefile   From time schedule   Transient data   From Surface

<< Back   Finish   Cancel   Help

# River Boundary

Edit Boundary Condition

Polylines: PLine0  
Zones: Polyline0.Zone0  
Points: Point 14

Show >>

Select how the attributes are defined

- Define for the entire zone.
- Define values at vertices (Linear Interpolation)
  - Just start and end points.
  - All vertices.

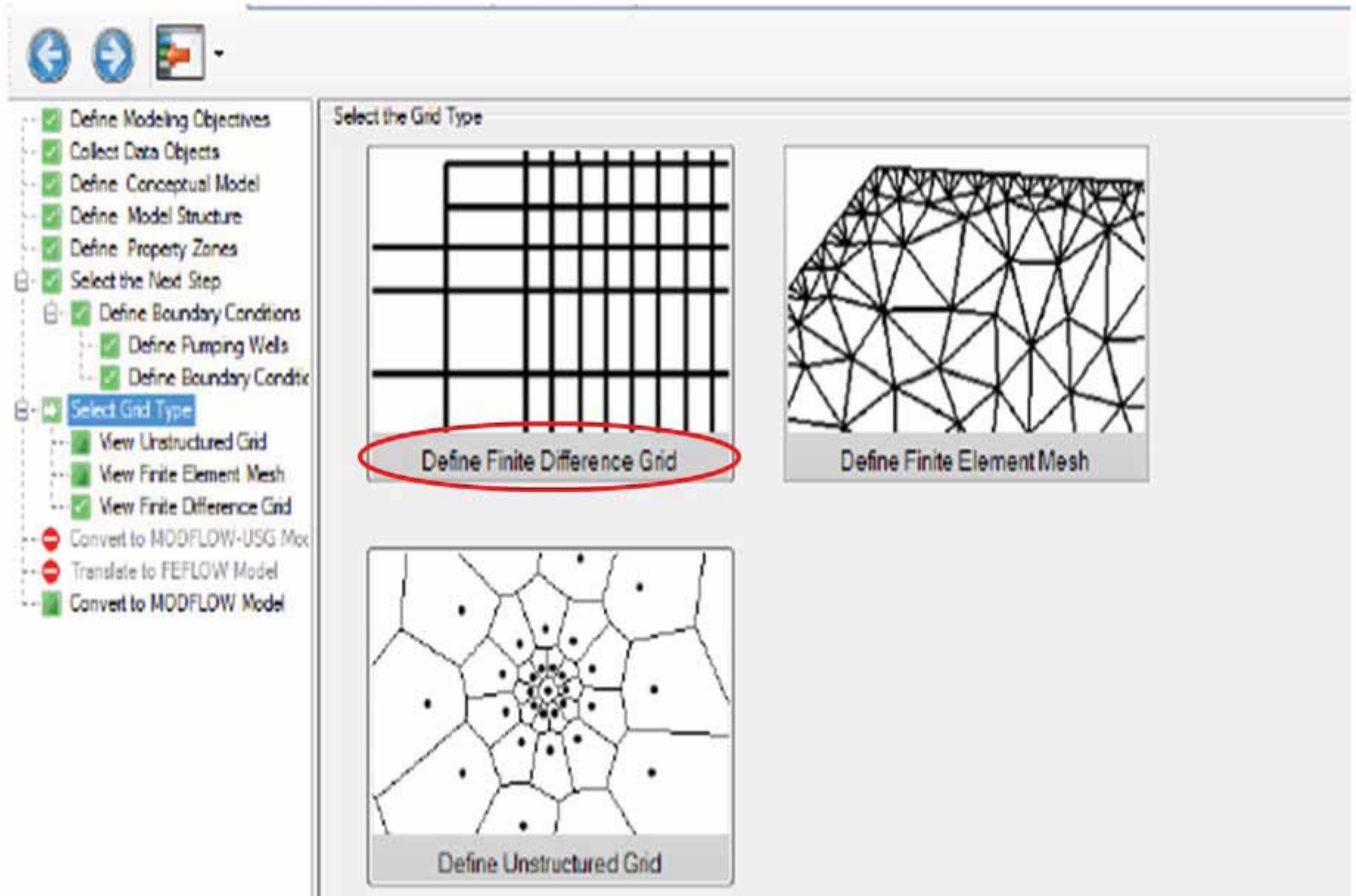
Use default leakance

Stage	Bottom	Leakance	Riverbed Thickness	River Width	Riverbed Conductivity
Static					
Constant					
335	333	K/\$RBTHICK	1	10	0.01

From 3D gridded   From shapefile   From time schedule   Transient data   From Surface

<< Back   Finish   Cancel   Help

# Model Grid Selection



The screenshot displays a software interface for model grid selection. On the left, a task list is visible, with the following items:

- Define Modeling Objectives
- Collect Data Objects
- Define Conceptual Model
- Define Model Structure
- Define Property Zones
- Select the Next Step
- Define Boundary Conditions
  - Define Pumping Wells
  - Define Boundary Conditions
- Select Grid Type** (highlighted)
- View Unstructured Grid
- View Finite Element Mesh
- View Finite Difference Grid
- Convert to MODFLOW-USG Model
- Translate to FEFLOW Model
- Convert to MODFLOW Model

The main area, titled "Select the Grid Type", shows three options:

- Define Finite Difference Grid**: A regular grid of squares. This option is highlighted with a red oval.
- Define Finite Element Mesh: A triangular mesh.
- Define Unstructured Grid: An irregular grid of polygons.



# Grid Design

Define Numerical Grid

Name  
NumericalGrid1

Define Horizontal Grid

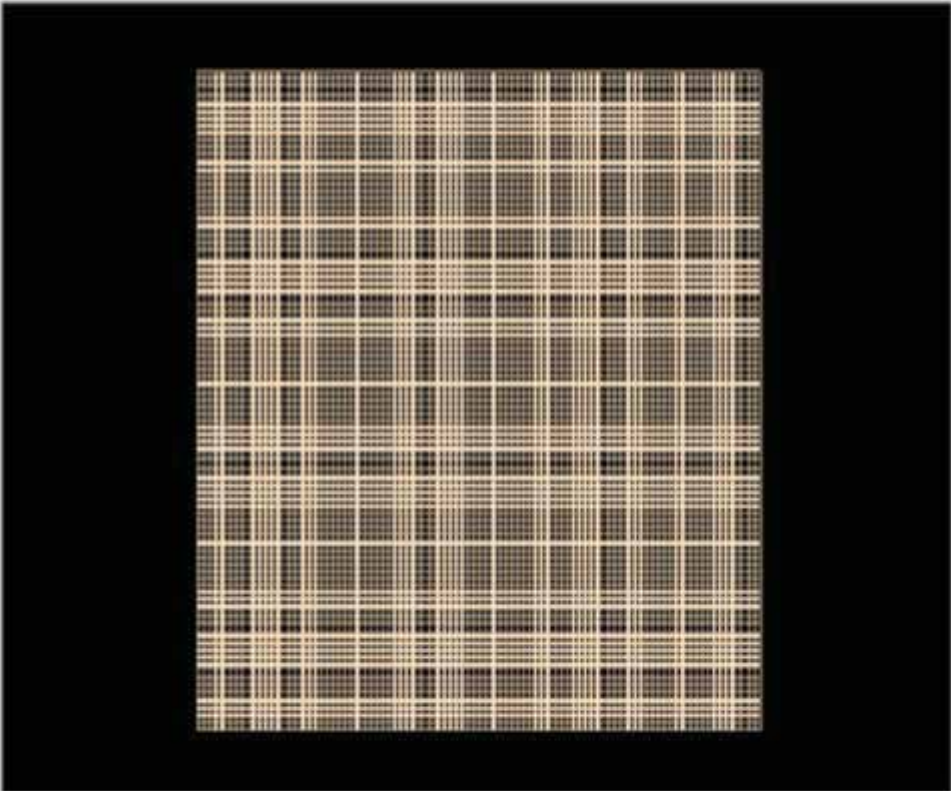
Rotation  
0

Grid Size

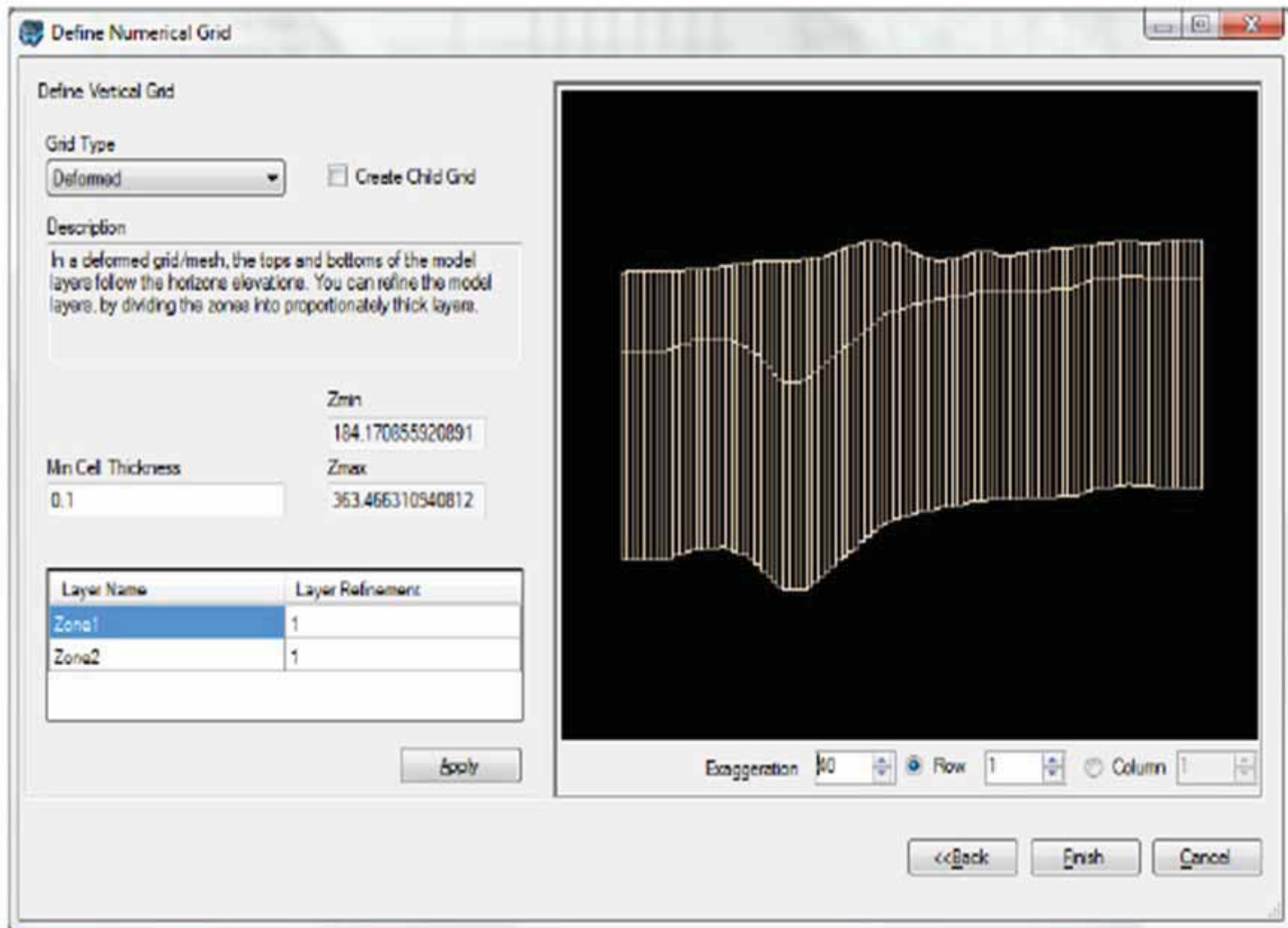
Rows	Cell Height
100	130.000
Columns	Cell Width
100	130.000

Grid Extents

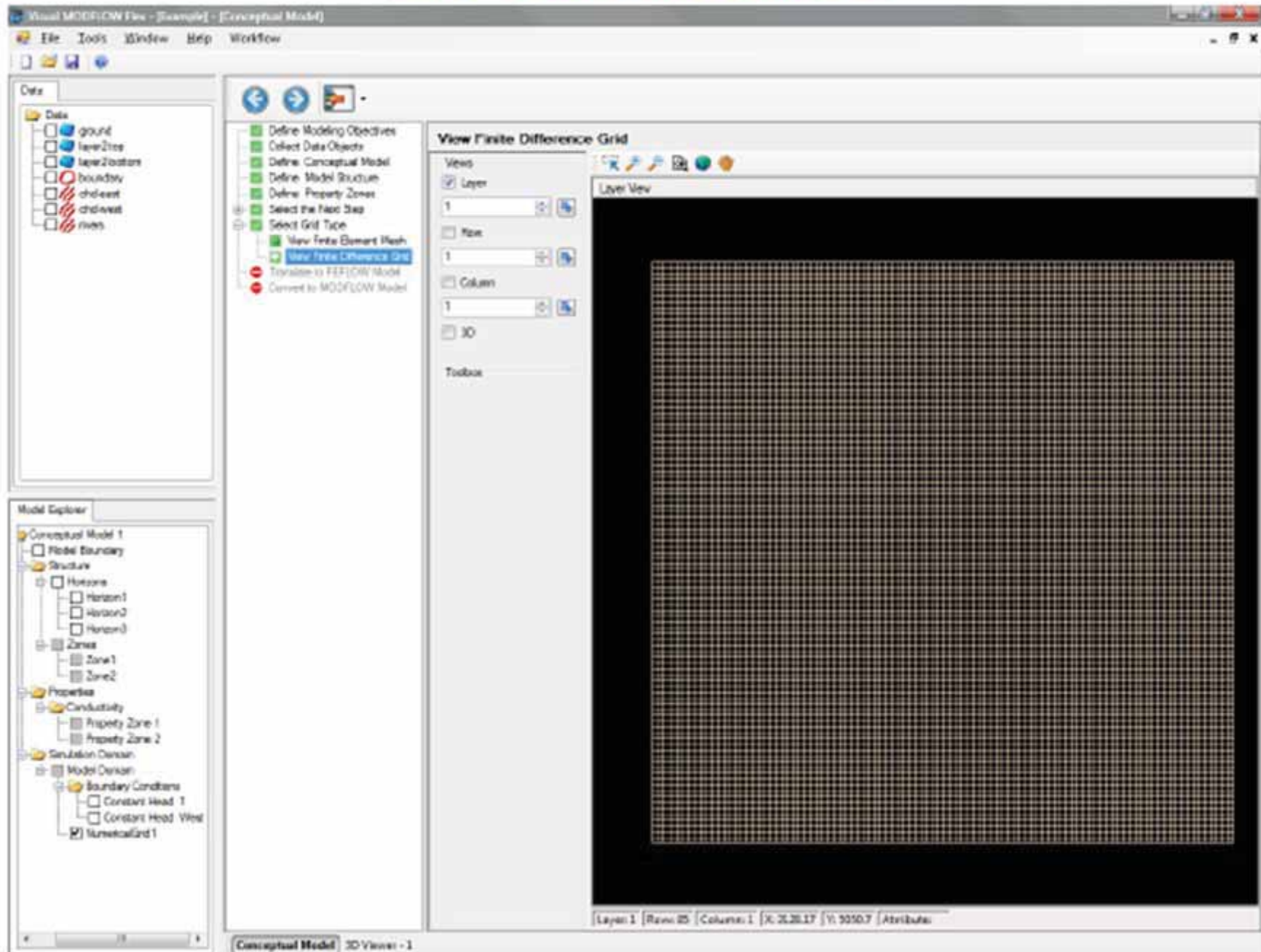
Xmin	Xmax	Width
3000.000	16000.000	13000.000
Ymin	Ymax	Height
3000.000	16000.000	13000.000



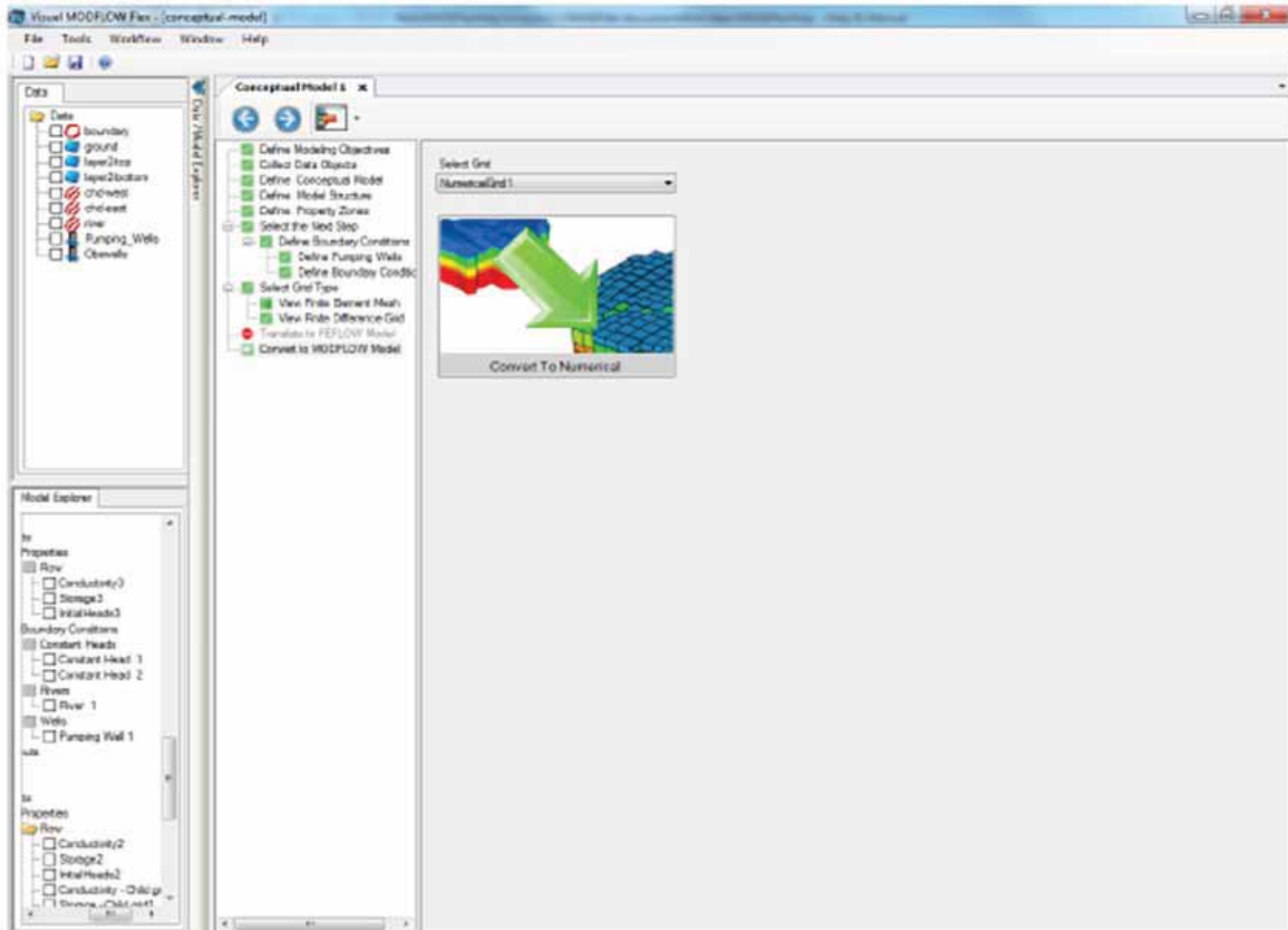
# Grid Type & Model Thickness



# 3D Visualization



# Conversion of Conceptual Model into Numerical Model



# Processing Conversion

The screenshot displays the Visual MODFLOW Flex interface during the conversion of a conceptual model to a numerical model. The window title is "Visual MODFLOW Flex - [conceptual-model]". The menu bar includes "File", "Tools", "Workflow", "Window", and "Help".

**Data Panel (Left):** Lists model components with checkboxes:

- boundary
- ground
- layer2top
- layer2bottom
- chd-west
- chd-east
- river
- Pumping\_Wells
- Observs

**Model Explorer (Bottom Left):** Shows properties for two model instances:

- to**
  - Flow
    - Conductivity3
    - Storage3
    - InitialHeads3
  - Boundary Conditions
    - Constant Heads
      - Constant Head 1
      - Constant Head 2
    - Rivers
      - River 1
    - Wells
      - Pumping Well 1
- nt**
  - Flow
    - Conductivity2
    - Storage2
    - InitialHeads2
    - Conductivity - Child gr
    - Storage - Child head 1

**Workflow Panel (Middle Left):** A tree view of the conversion process:

- Convert conceptual model to numerical model
  - Define Properties
  - Define Boundary Conditions
  - Select the Next Step
  - Define Observation Zones
  - Define Observation Wells
  - Define Zone Budget Zones
  - Define Patches
  - Select Run Type
    - PEST Run
    - Single Run
  - Transition
  - Run Numerical Engine
  - View Results
    - View Charts
    - View Maps

**Main Panel (Right):** Titled "Converting conceptual model to numerical model", it displays a log of conversion steps:

- Converting Properties:
  - Converting property for conductivity, the layer number is: 1
  - Converting property for conductivity, the layer number is: 2
  - Converting property for storage, the layer number is: 1
  - Converting property for storage, the layer number is: 2
  - Converting property for initial\_heads, the layer number is: 1
  - Converting property for initial\_heads, the layer number is: 2
- Conductivity
- Storage
- InitialHeads
- Converting Boundary Conditions:
  - Constant Head 1
  - Constant Head 2
- River 1
- Pumping Well 1

Conceptual model to numerical model conversion has completed

# View & Edit Properties

The screenshot displays a software interface with the following components:

- Top Bar:** Contains window titles "Conceptual Model 1" and "NumericalGrid1-Run3", and navigation icons (back, forward, home).
- Left Panel (Workflow Tree):** A tree view of steps including "Convert conceptual model to num", "Define Properties" (highlighted), "Define Boundary Conditions", "Select the Next Step", "Define Observation Wells", "Define Zone Budget Zones", "Define Particles", "Select Run Type" (with sub-items "PEST Run" and "Single Run"), "Translate", "Run Numerical Engines", "View Results", "View Charts", and "View Maps".
- Inspect/Define Properties Panel:** A central panel with the following sections:
  - Views:** Checkboxes for "Layer" (checked), "Row", "Column", and "3D". Below each checkbox is a numeric input field set to "1".
  - Toolbox:** A series of dropdown menus for "Conductivity", "Zone", "Database", and "Assign".
  - Legend:** A table with two columns: "Color" and "Zones".
- Layer View:** A large grid view showing a light-colored grid pattern on a dark background.
- Bottom Bar:** A status bar with labels for "Layer:", "Row:", "Column:", "HCoord:", and "VCoord:".

Color	Zones
	1
	2

# Property Views

Views

Layer

1

Row

1

Column

1

3D

Toolbox

Initial Heads

Zone

Database

Assign

# View & Edit Boundary Condition

The screenshot displays a software interface for defining boundary conditions. On the left, a project tree shows the current step, "Define Boundary Conditions", highlighted in blue. The main window is titled "Inspect/Define Boundary Conditions" and features a "Layer View" of a grid. A blue boundary line is visible on the grid, flanked by red vertical lines. The interface includes a "Views" panel with options for Layer, Row, Column, and 3D. A "Toolbox" contains buttons for "Constant Head", "Edit", "Assign", and "Database". At the bottom, a status bar shows fields for "Layer:", "Row:", "Column:", "HCoord:", "VCoord:", and "Attribute:".

**Inspect/Define Boundary Conditions**

Views

- Layer
- Row
- Column
- 3D

Toolbox

- Constant Head
- Edit
- Assign
- Database

Layer View

Layer: | Row: | Column: | HCoord: | VCoord: | Attribute:



# Creation of Zones for Zone Budget

Navigation icons: back, forward, and a small map icon.

- Define Modeling Objectives
  - Define Numerical Model
    - Create Grid
    - Import Model
  - View/Edit Grid
  - Define Properties
  - Define Boundary Conditions
  - Select the Next Step
    - Define Observation Wells
    - Define Zone Budget Zones
    - Define Particles
  - Select Run Type
    - PEST Run
    - Single Run
  - Translate
  - Run Numerical Engines
  - View Results
    - View Charts
    - View Maps

### Choose the Next Step

Select Run Type

Define Particles

Define Zone Budget Zones

Time (d)	Head (m)
0	213.1
30	214.5
60	213.7
90	212.4
120	212.4

Define Observation Wells

# Select Run Type

The screenshot shows a software interface for selecting a run type. On the left is a sidebar with a tree view of steps. The main area is titled "Select Run Type" and contains a table of engines and a large "PEST" button.

**Select Run Type**

Engine	Run
MODFLOW 2000	<input checked="" type="checkbox"/>
MODPATH	<input checked="" type="checkbox"/>
Zone Budget	<input type="checkbox"/>
MT3DMS	<input type="checkbox"/>

Single Run

PEST Run

**PEST**

PEST Run

# Select Modflow Engine

**Compose Engines**

TypeOfEngine	Run	Description
MODFLOW-2000	<input type="checkbox"/>	USGS MODFLOW 2000 from SWS
MODFLOW-2005	<input checked="" type="checkbox"/>	USGS MODFLOW 2005 from SWS
MODFLOW-LGR	<input type="checkbox"/>	USGS MODFLOW-LGR from SWS
ZONEBUDGET	<input type="checkbox"/>	ZONEBUDGET
MODPATH	<input type="checkbox"/>	MODPATH

# Translate Model

The screenshot shows the 'WELLHEAD-CAPTURE-ZONE grid-Run' application window. The top toolbar contains several icons, with the 'Translate' icon (a blue play button) highlighted by a red rectangle. The left sidebar displays a task list with the following items:

- Define Modeling Objectives
  - Define Numerical Model
    - Create Grid
    - Import Model
  - View/Edit Grid
  - Define Properties
  - Define Boundary Conditions
  - Select the Next Step
    - Define Observation Wells
    - Define Zone Budget Zones
    - Define Particles
  - Select Run Type
    - PEST Run
    - Single Run
  - Translate** (highlighted)
  - Run Numerical Engines
  - View Results
    - View Charts
    - View Maps

The main window is titled 'Translate' and displays the following settings:

- General
  - Settings
- MODFLOW-2000
  - Settings
  - Time Steps
  - Solvers
  - Recharge and EVT
  - Lake
  - Layers
  - Rewetting
  - Initial Heads
- MODPATH
  - Discharge Options
  - Time Format

The 'General' section is expanded, showing the following table:

General	
Output Folder	D:\Documents\Visual MODFLOW Flex\Projects\sample.E
Start Date	10/5/2005

# Model Translation in Progress

The screenshot displays a software interface for model translation, titled "WELLHEAD-CAPTURE-ZONE grid-Run". The interface is divided into several panels:

- Data Panel:** Lists various data sources, including "wellhead-capture-zone Elevation #1" through "#7", "VWd Imported Wells", and "Head Observation Wells".
- Model Explorer:** Shows a hierarchical tree structure. Under "Simulation Domain", "Model Domain" is expanded to show "Boundary Conditions" (with "Wells" checked) and "WELLHEAD-CAPTURE-ZONE grid". The "Run" section is also expanded, showing "Inputs" (Properties, Transport, Boundary Conditions, Zone Budget, Particles, Observations) and "Outputs".
- Workflow Pane:** A central pane showing a sequence of steps: "Define Modeling Objectives", "Define Numerical Model", "Create Grid", "Import Model", "View/Edit Grid", "Define Properties", "Define Boundary Conditions", "Select the Next Step", "Define Observation Wells", "Define Zone Budget Zones", "Define Particles", "Select Run Type", "PEST Run", "Single Run", "Translate", "Run Numerical Engine", "View Results", "View Charts", and "View Maps".
- Translate Log Details:** A window showing a log of translation activities. The log entries include timestamps and descriptions of translation steps for various stress periods and packages, such as "Translate for Stress Period # 2", "RIV Package translator: Initialize", "CHD Package...", "Schlumberger Water Services", "Recharge Package translator: Initialize", and "GHB Package...".

# Run Engine

The screenshot shows the 'Run Engine' software interface. On the left is a task tree with the following items: Define Modeling Objectives (checked), Define Numerical Model (checked), Create Grid (checked), Import Model (checked), View/Edit Grid (checked), Define Properties (checked), Define Boundary Conditions (checked), Select the Next Step (checked), Define Observation Wells (checked), Define Zone Budget Zones (checked), Define Particles (checked), Select Run Type (checked), PEST Run (checked), Single Run (checked), Translate (checked), Run Numerical Engines (highlighted in blue), View Results (unchecked), View Charts (unchecked), and View Maps (unchecked). The top toolbar contains navigation arrows and 'Run' and 'Stop' buttons. The main window is titled 'Run Numerical Engines' and shows the execution of 'MODFLOW-2000 MODPATH-2000(B)'. The output text is as follows:

```
U. S. GEOLOGICAL SURVEY MODULAR FINITE-DIFFERENCE GROUND-WATER FLOW MODEL
Version 1.19.01 03/25/2010 Prec:single, Reg:DOS
MODFLOW-2000
OpenMP-parallelized using 2 CPU
Using NAME file: D:\Documents\Visual MODFLOW Flex\Projects\sample.data\MODFLOW
\WELLHEAD-CAPTURE-ZONE grid\Run\MODFLOW-2000\WELLHEAD-CAPTURE-ZONE.modflow.in
Run start date and time (yyyy/mm/dd hh:mm:ss): 2013/10/30 12:32:21
Period: 1 Step: 2

CGSTAB-P linear solver

Programmed by Michael S. Obrecht
SIBORG SYSTEMS Inc.

24 Combermere Cres., Waterloo
Ontario, Canada N2L 5B1
Phone (519) 888-9906
FAX (519) 725-9522

Iterations: Outer: 1 Inner: 46 Residual: 8.981E-04 Max Change: -2.426E+01
Convergence: Residual: 8.9811982E-04 Max Change: -24.25531
```

# View Results

The screenshot displays the Visual MODFLOW Flex software interface. The main window title is "Visual MODFLOW Flex - [conceptual-model]". The menu bar includes "File", "Tools", "Workflow", "Window", and "Help".

**Data Panel (Left):** Lists model components such as boundary, ground, layer2-top, layer2-bottom, chd-west, chd-east, river, Pumping\_Wells, and Observels.

**Workflow Panel (Middle-Left):** Shows a sequence of steps for running a numerical grid. The "View Results" step is currently selected and highlighted in blue. Other steps include: Convert conceptual model to num, Define Properties, Define Boundary Conditions, Select the Next Step, Define Observation Zones, Define Observation Wells, Define Zone Budget Zones, Define Packages, Select Run Type (PEST Run, Single Run), Translate, Run Numerical Engine, View Charts, and View Maps.

**Model Explorer (Bottom-Left):** A tree view for the "NumericalGrid1" model, showing sub-categories like Run, Inputs (Properties, Boundary Condition, Rivers, Wells, River, Observation Wells), Outputs, and Run1 (Inputs, Outputs).

**View Results Panel (Right):** Contains two interactive buttons: "View Maps" and "View Charts".

**View Maps:** A thumbnail image showing a 2D contour map of a subsurface domain with a color gradient from red (high head) to green (low head).

**View Charts:** A thumbnail image showing a 1D plot of data points (blue circles) along a diagonal axis, with a red triangle at the bottom left.

# View Maps

The screenshot displays the Visual MODFLOW Flex software interface. The main window is titled "Visual MODFLOW Flex - [conceptual-model]". The interface is divided into several panels:

- Data:** A tree view on the left showing model components like boundary, ground, layer2top, layer2bottom, chd-west, chd-east, river, Pumping\_Wells, and Observs.
- Conceptual Model 1 / NumericalGrid1-Run1:** A central panel with a workflow tree. The "View Maps" step is highlighted in blue.
- Model Explorer:** A tree view at the bottom left showing the model structure, including Run, Inputs, Properties, Boundary Condition, Constant Head, Rivers, Wells, and Outputs.
- View Maps:** A central panel with a "Views" section (Layer, Row, Column) and a "Toolbox Legend" table.
- Layer View:** A large map area showing a color-coded head distribution. The map shows a gradient from blue (low head) on the left to red (high head) on the right.

The "Toolbox Legend" table is as follows:

Color	Heads (ft)
Dark Blue	325.0000
Blue	328.1429
Light Blue	331.2857
Green	334.4286
Yellow-Green	337.5714
Yellow	340.7143
Orange	343.8571
Red	347.0000

At the bottom of the Layer View, the status bar indicates: Layer 1 | Row:48 | Column:22 | X: 5735.28 | Y: 9882.47 | Heads: 330.6457



