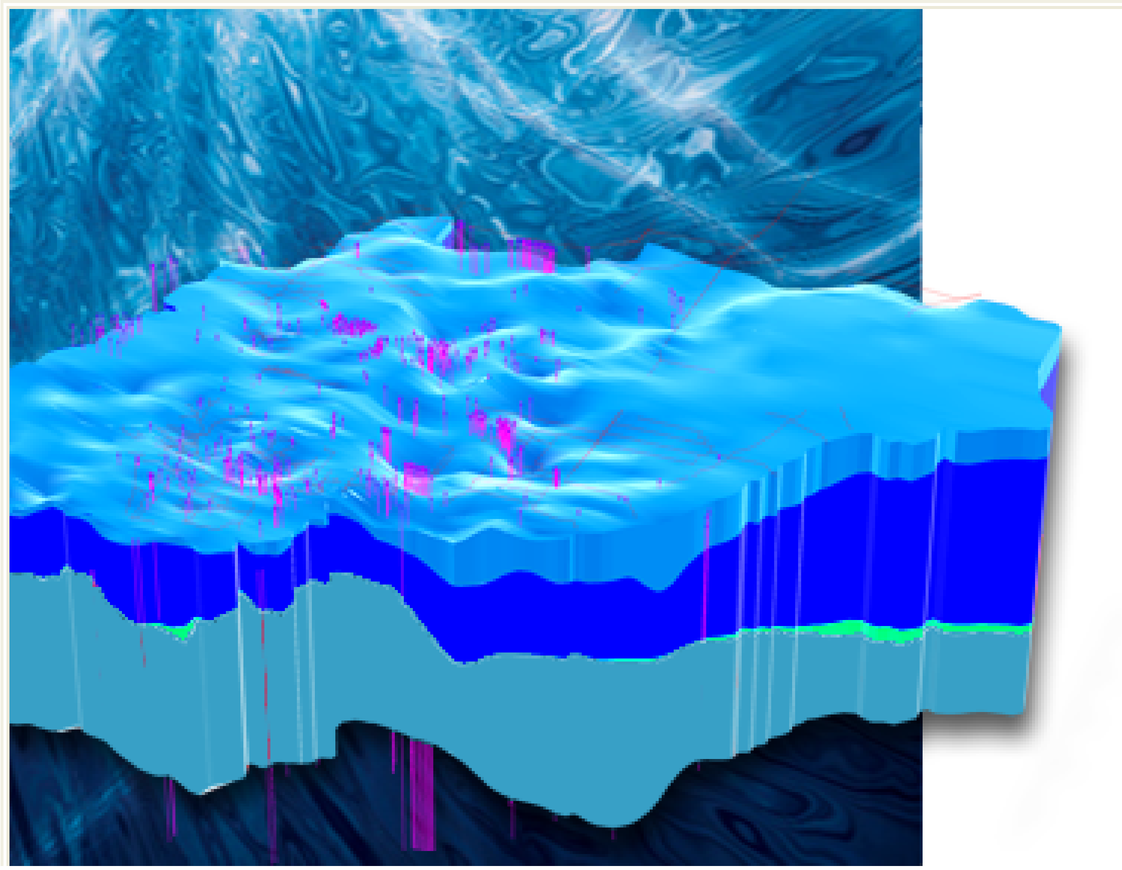
Tutorial:

Importing VMOD/MODFLOW Models



Visual MODFLOW Flex 6.1

Integrated Conceptual & Numerical
Groundwater Modeling Software

Waterloo Hydrogeologic, Inc.
219 Labrador Drive, Suite 201
Waterloo, ON N2K 4M8
CANADA

www.waterloohydrogeologic.com

Published: July 2019 in Waterloo, Canada

1 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:

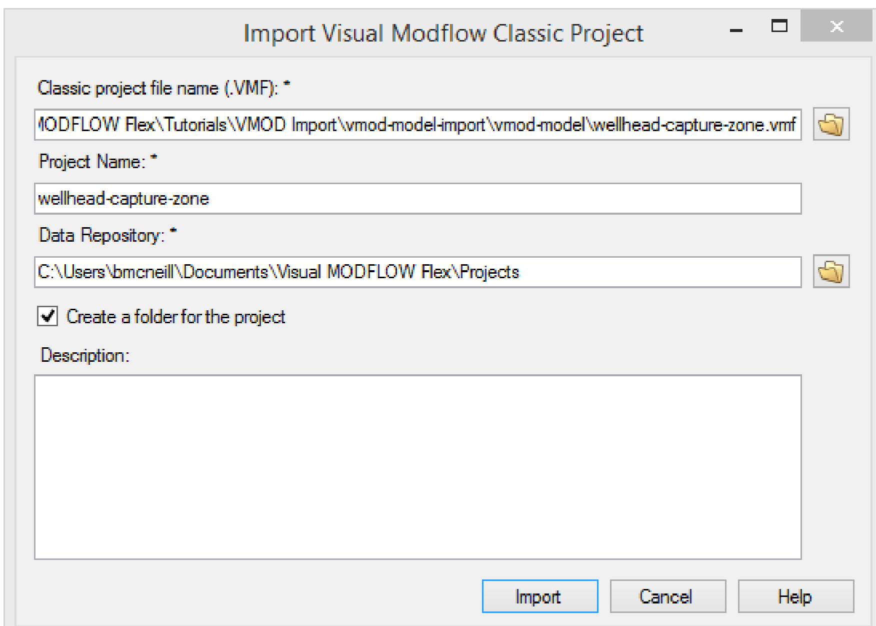
- Flow Engines (MODFLOW-96)
- Flow Packages (ETS1, MNW, STR)
- Transport Engines (MT3D99, PHT3D)
- MGO

Importing a Visual MODFLOW Classic Project

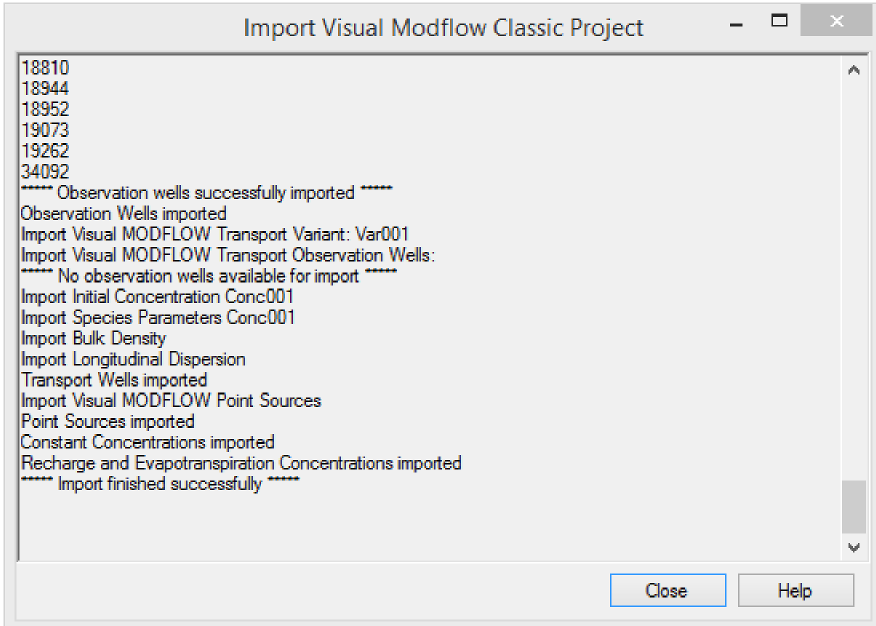
The release of Visual MODFLOW Flex v5.0 introduced a new method of importing Visual MODFLOW Classic projects. This new method simplifies the process to import an existing Visual MODFLOW Classic project. The new method combines the 'Create a new project' and 'import model files' processes into a single step. This section describes the new procedure for importing Visual MODLFOW 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).
  - o 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:

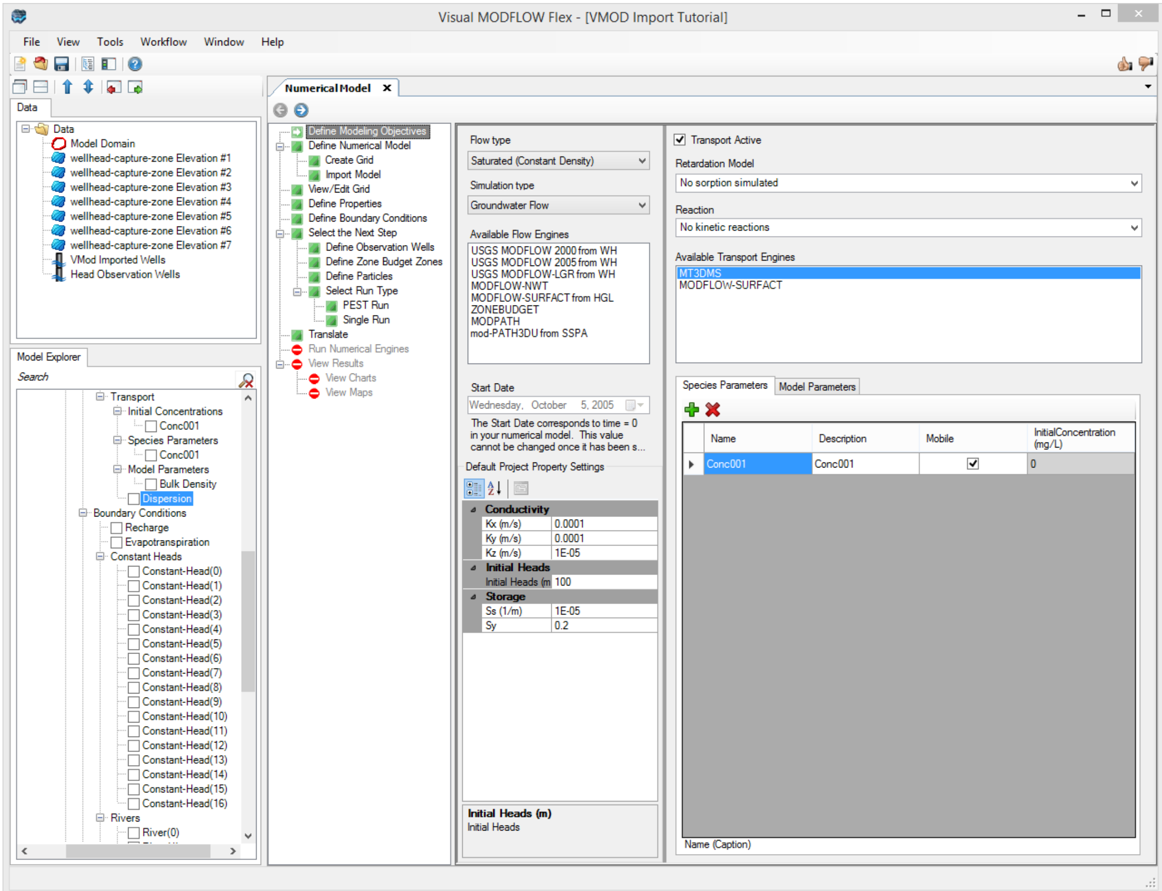


- Enter a project name for the Visual MODFLOW Flex project (by default it will enter the same project name as the Visual MODFLOW Classic project).
  - o In this example the project has been named 'VMOD Import Tutorial'
- Select a data repository for the Visual MODFLOW Flex project, select whether to 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:



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:



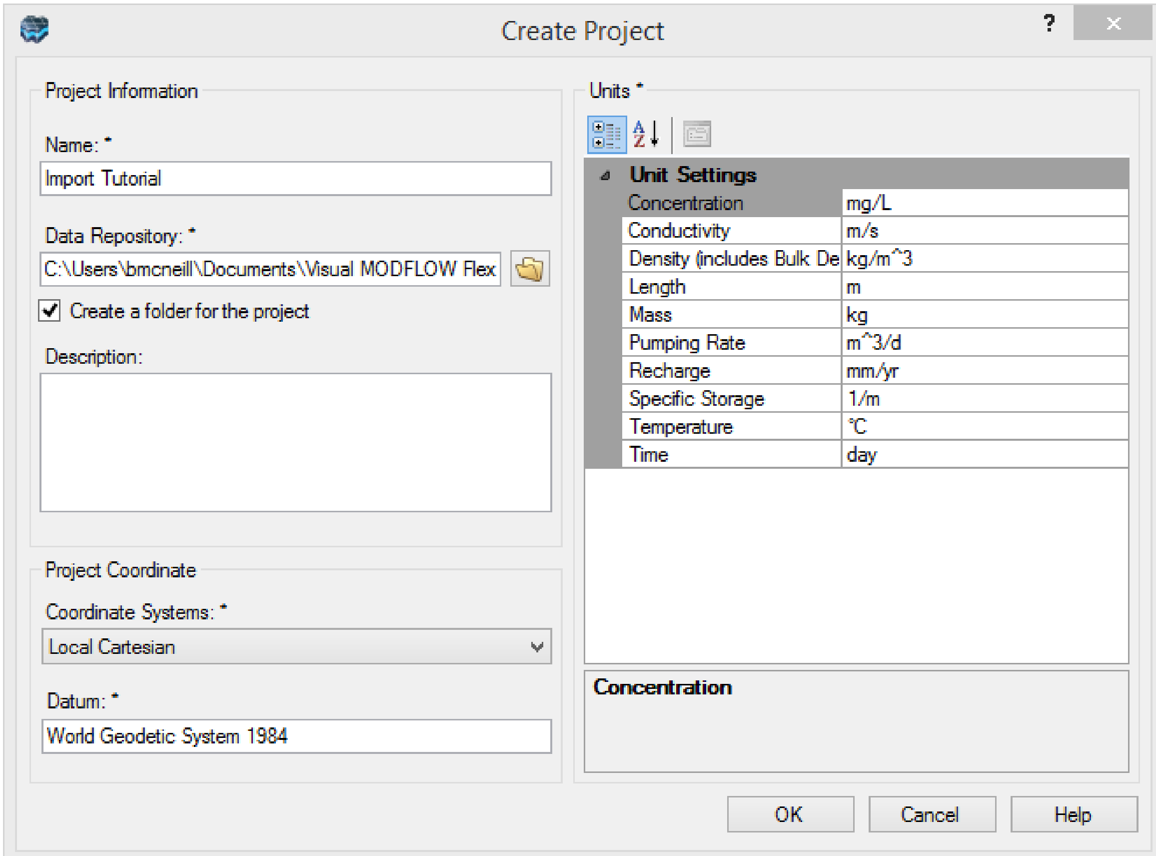
You can now continue with the regular numerical modeling workflow by proceeding to the section of this tutorial titled View/Edit Grid.

Creating the Project and Import Model Files

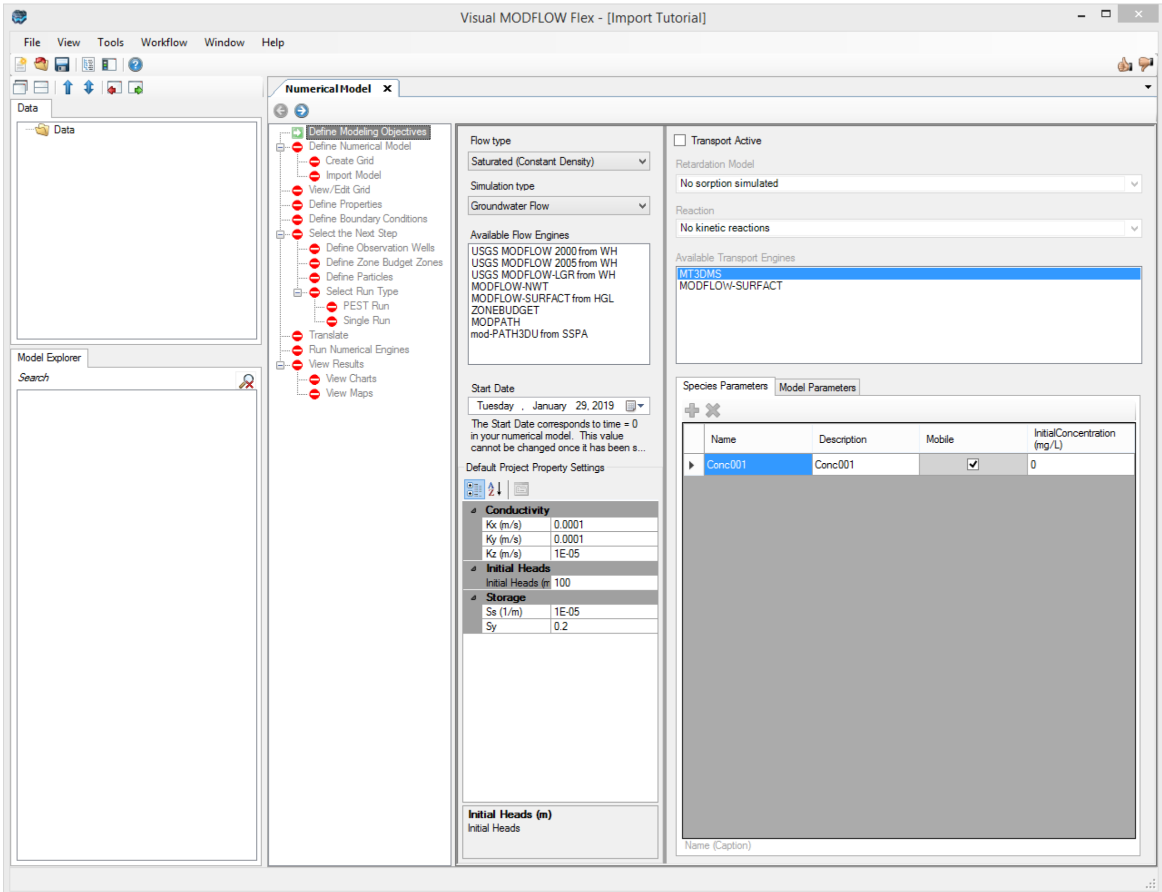
This section describes the procedure for importing MODFLOW project files (developed independently of the Visual MODFLOW Classic/Flex interface) into the Visual MODFLOW Flex interface. This procedure can also be used to import Visual MODFLOW Classic projects, but the procedure outlined in the previous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



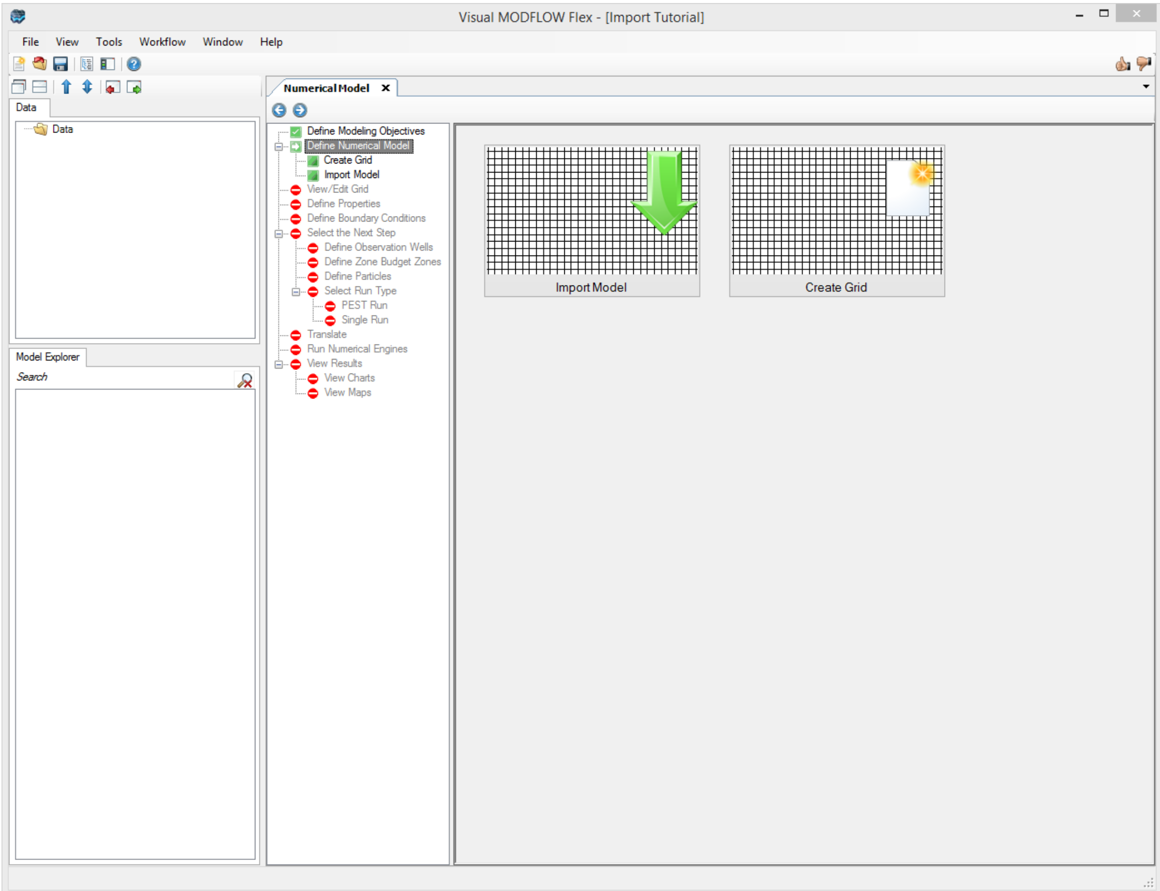
- Click **[OK]**. The workflow selection screen will appear.
- Select **[Numerical Modeling]** and the Numerical Modeling workflow will load.
- In this step, you define the objectives of your model and the default parameters.



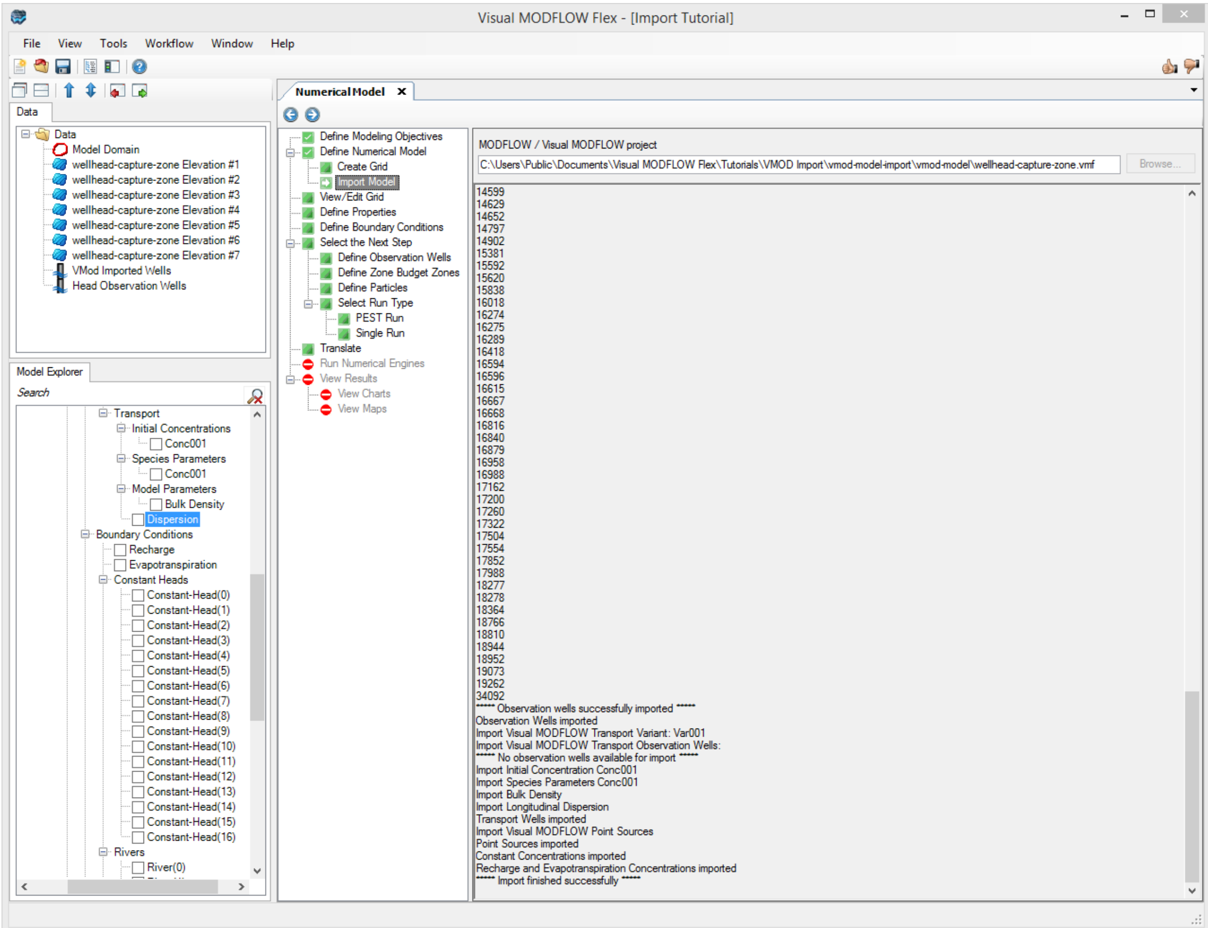
- For the Start Date, expand the date picker, and select **1/1/2000**

  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.
  - o 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))
  - o 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.



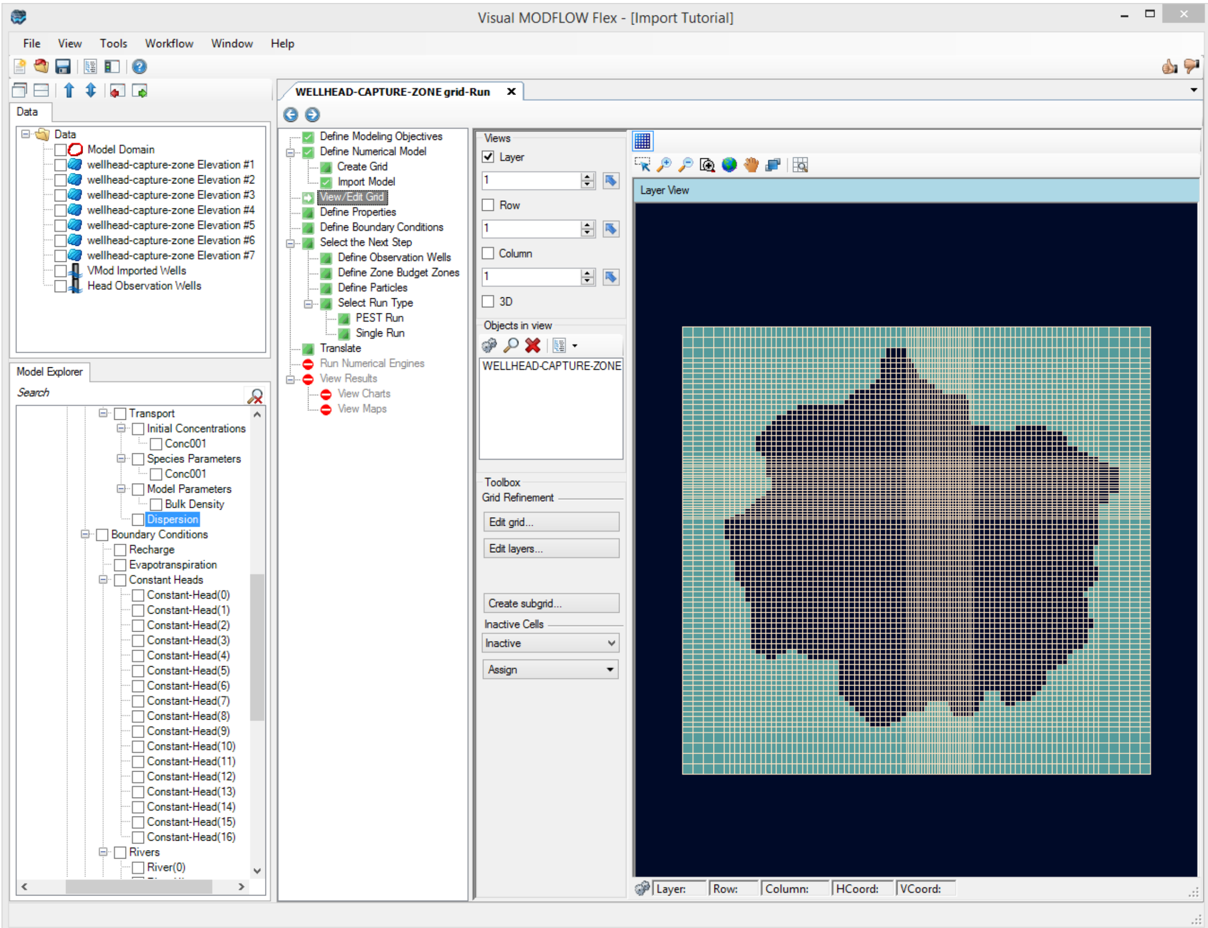
- The status of each model element import is shown in the main window; any detected errors will be shown here.
- After the import, you will see the **'Model Explorer'** is populated in the bottom left corner of the screen; from here, you can show/hide different model inputs/outputs.

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:

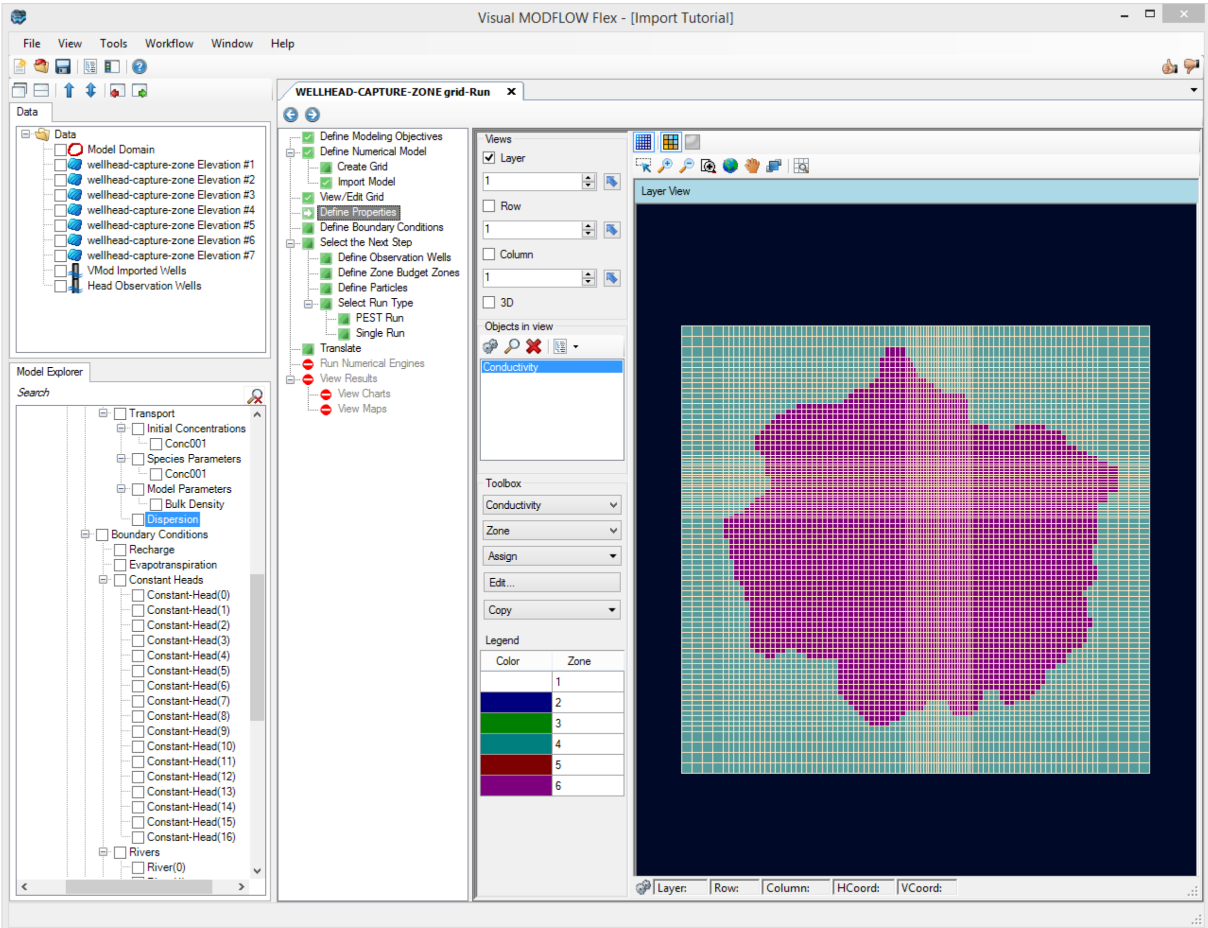


- Under **'Views'**, select the various views you want to see in the Flex viewer; VMOD Flex allows you to simultaneously show a layer, row, column and 3D Views. Place a check box beside the desired view and it will appear on the screen.
- Adjust a specific layer, row, or column using the up/down arrows. 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.

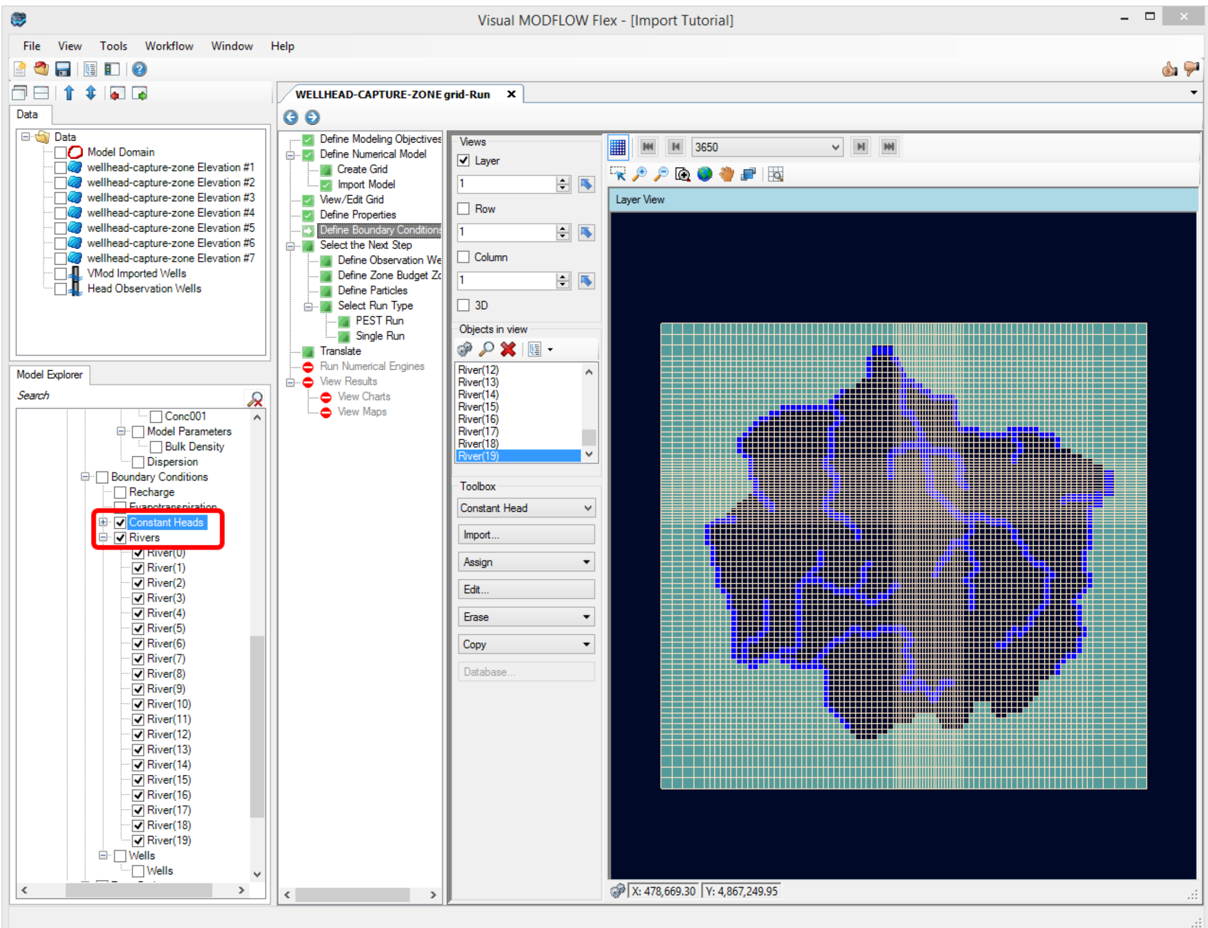
The display tools will allow you to switch from discrete cells rendering to color shading/contours (note, this is available only when you do attribute rendering, and not when you are rendering by ZoneID)

	Show/hide grid lines
	Show as cells
	Show as Surface

- 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) 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 (eg. River), then this set of boundary condition cells will become visible.

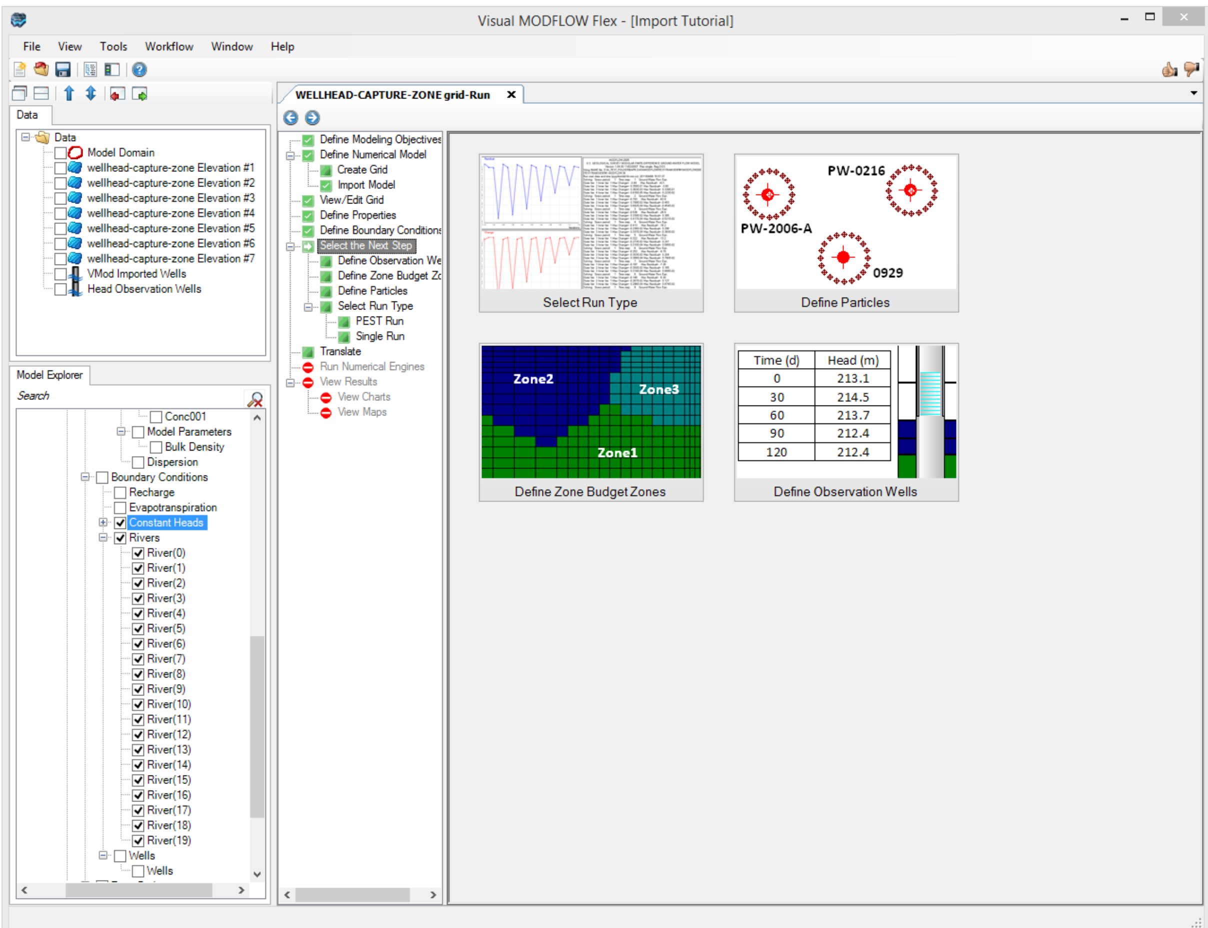


- From the toolbox, select the desired boundary condition group (Constant Head, Rivers, etc..)
- Then select **[Edit...]**.
- Click on a cell that belongs to the boundary you are interested in; a dialog will appear where you can see the parameters for all cells in that boundary. If you want to switch 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.

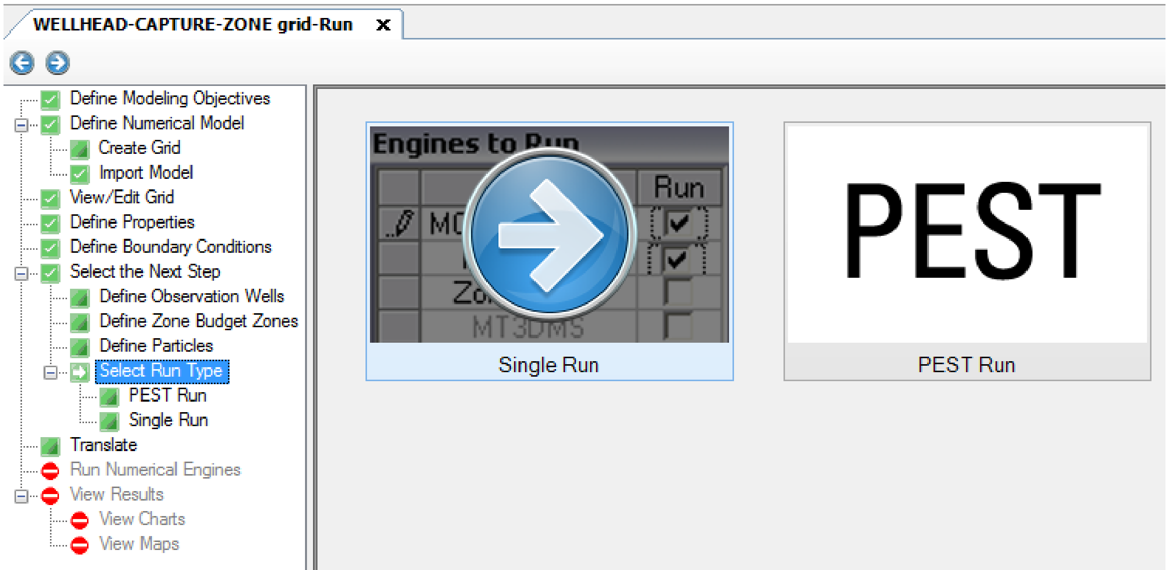
- For Recharge or Evapotranspiration, you need to select this Boundary Condition type from the list under the Toolbox, and the Recharge/EVT zonation will appear. A **[Database]** button will be enabled allowing you to view the zonations.
- You can also assign new boundary conditions to the model using the **[Assign]** options; for more details, please see Define Boundary Conditions
- Click **[ ]** (Next Step) to proceed. You will arrive at the "Select the Next Step" options screen

Proceed to Run or Define Optional Model Elements

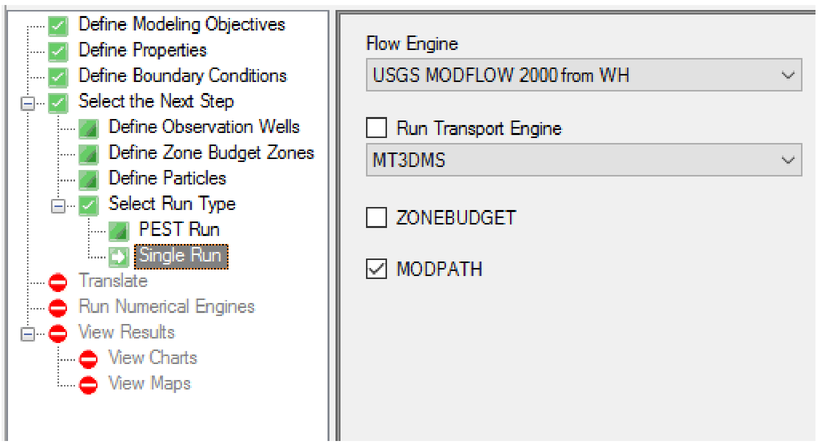
- You will arrive at a choice screen; here you can proceed to some of the “non-essential” inputs for the model, such as Zone Budget Zones, Particle Tracking, and Observation Wells. Or, you can proceed to Running the simulation.



- 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.

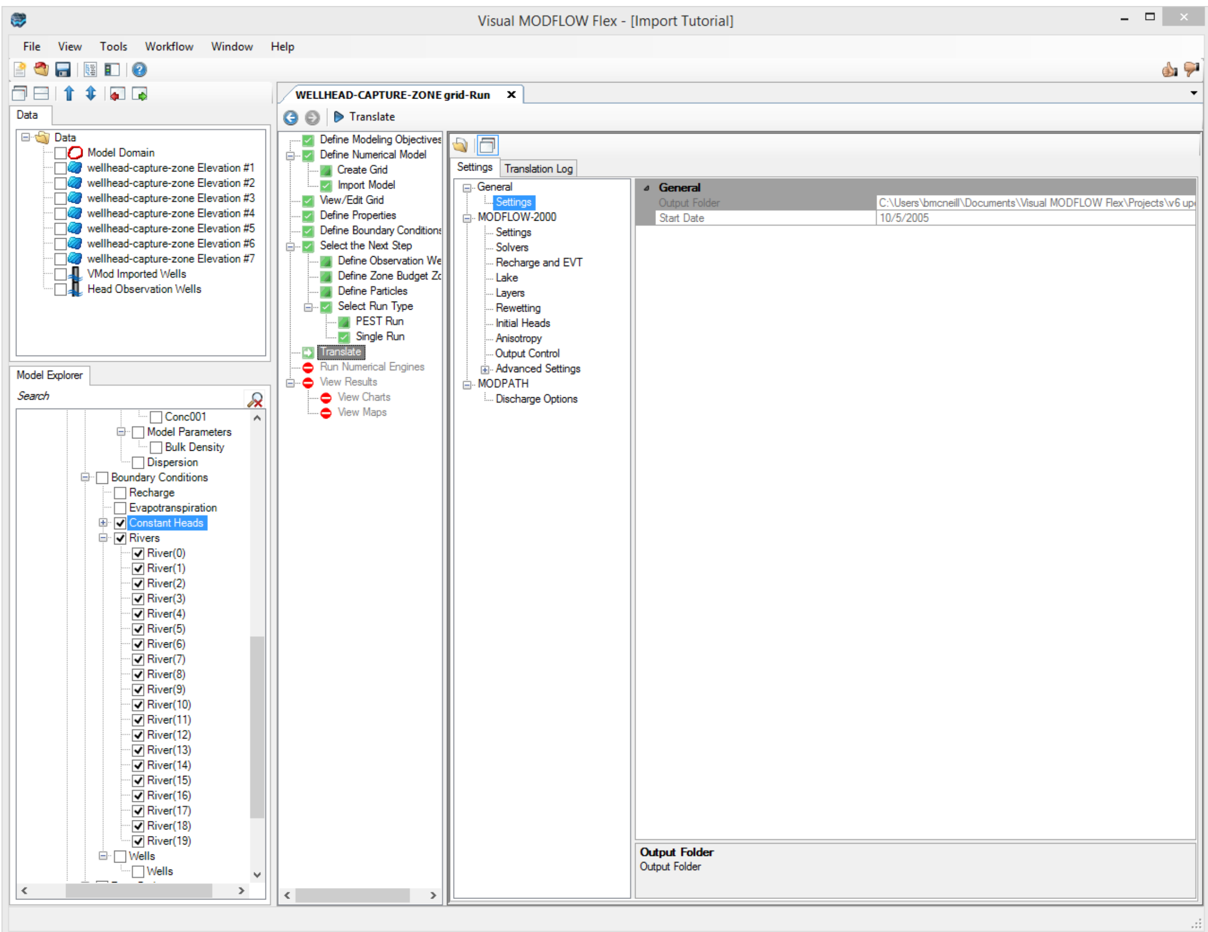
Please Note: *****De-select MT3DMS engine.***** since there are no transport inputs defined for this model.

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

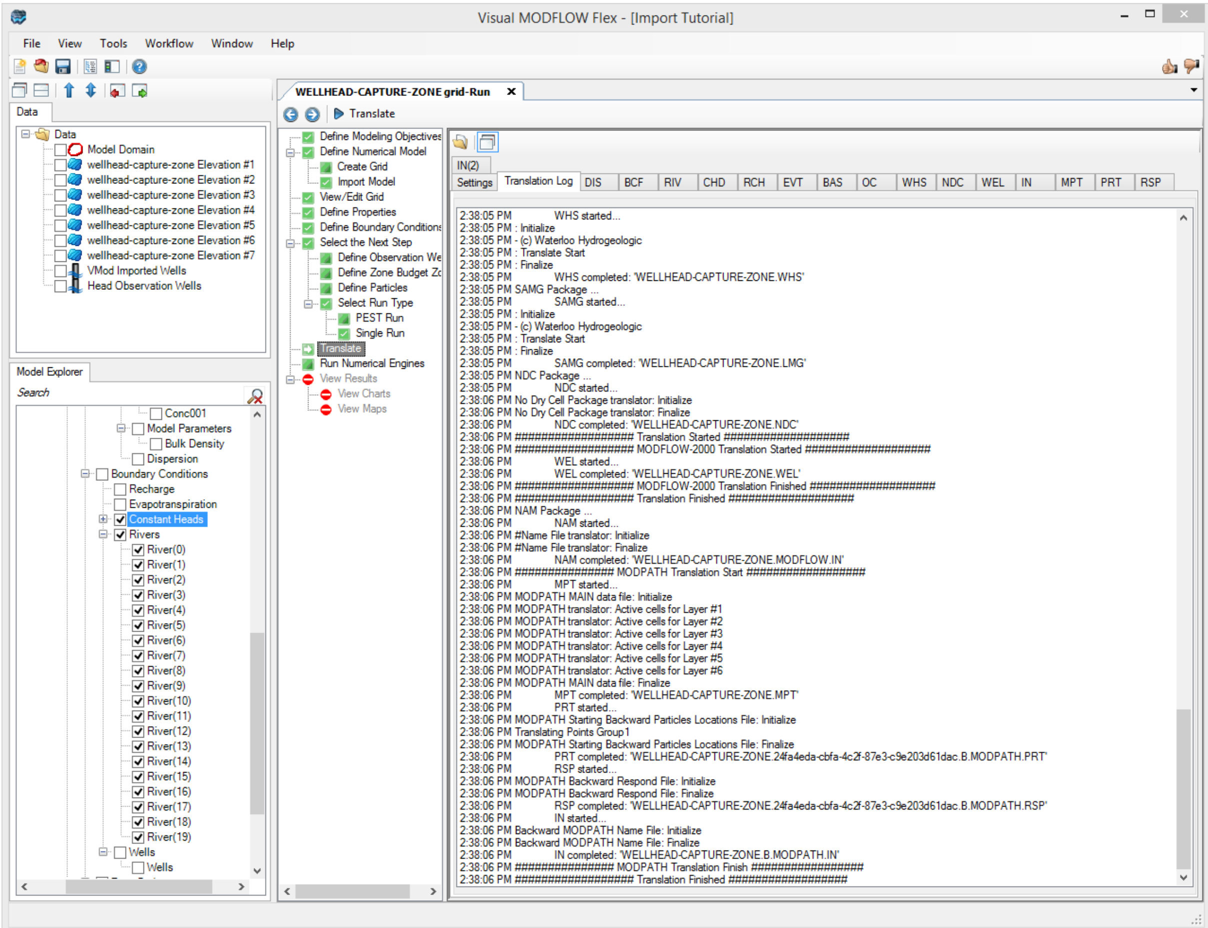
Translate Packages

- 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.

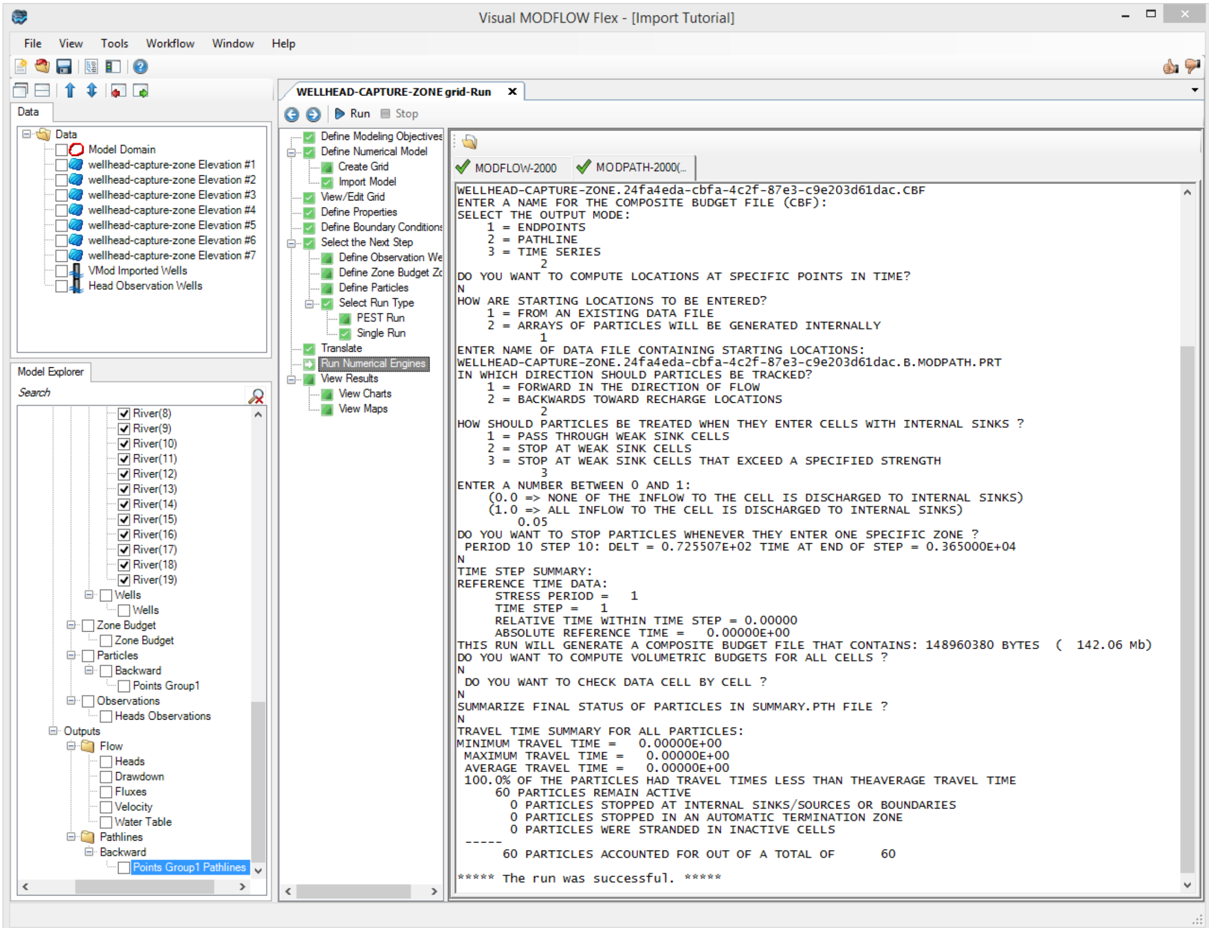


- Click the [] (Next step) button to proceed. You will arrive at the “Run Engines Step”.

Run Engines

- Click the [

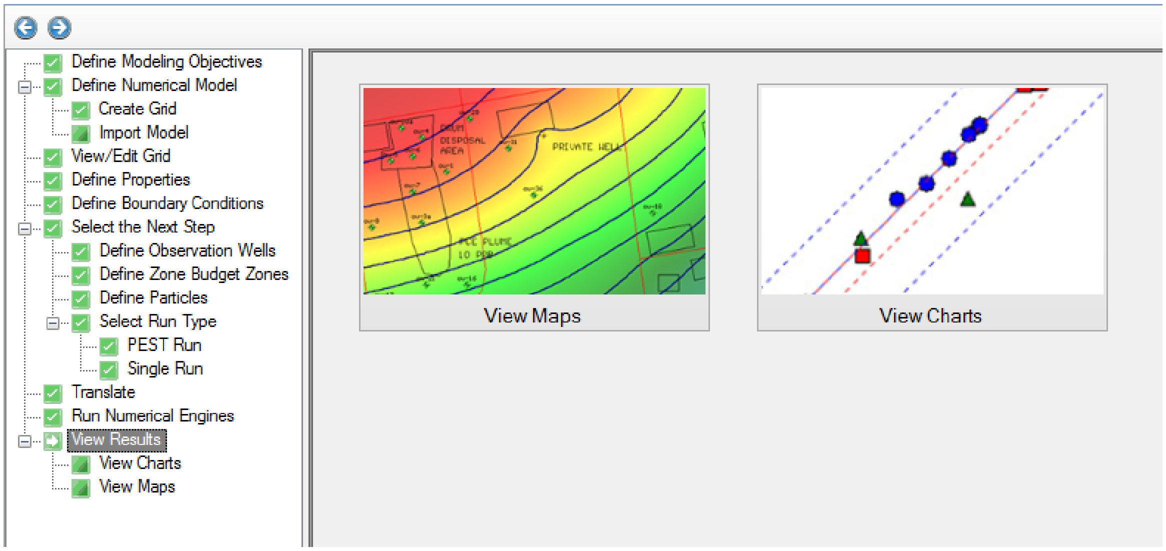
] button on the main workflow toolbar to start running the engines. You will see the Engine progress in the scrolling window.



- 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.

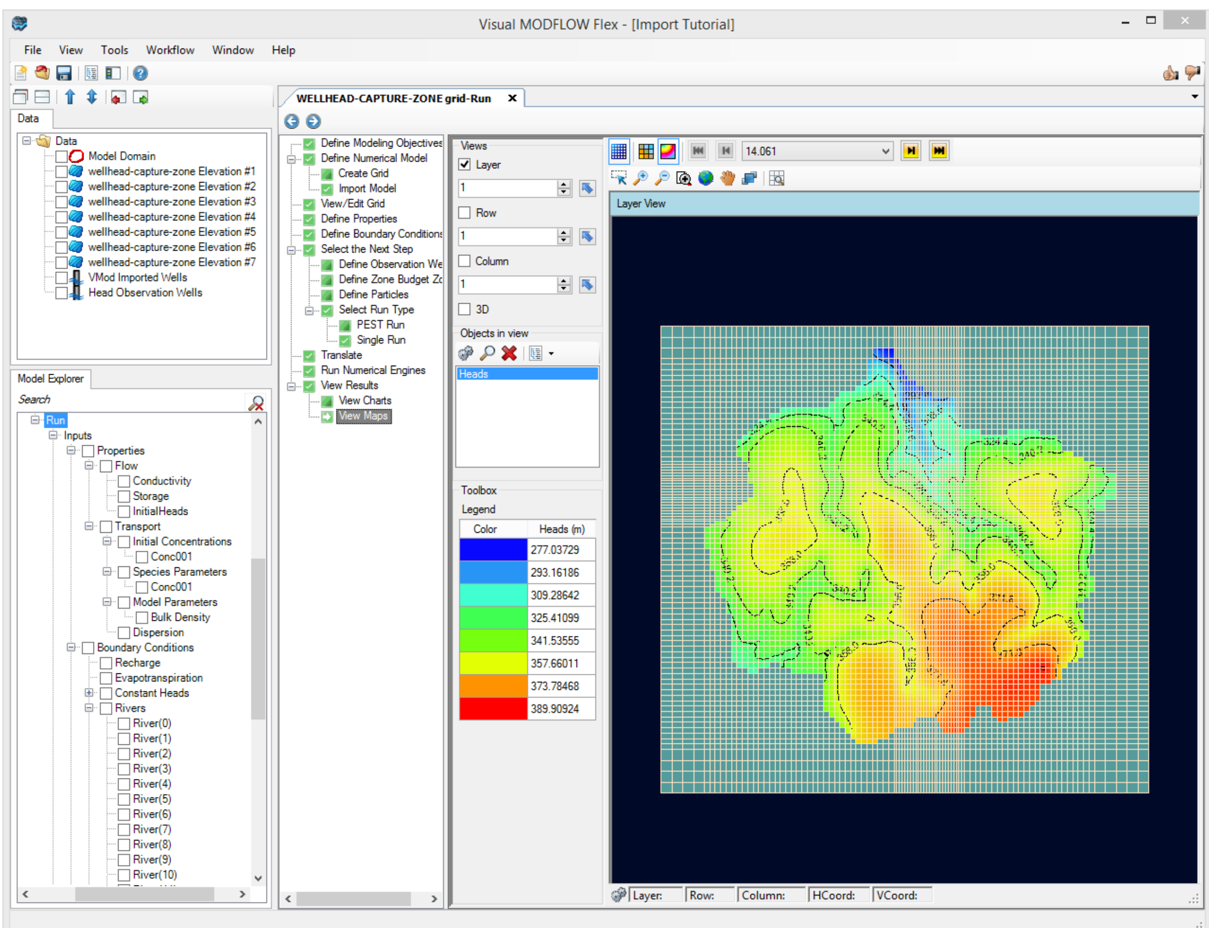
View Results

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



View Maps

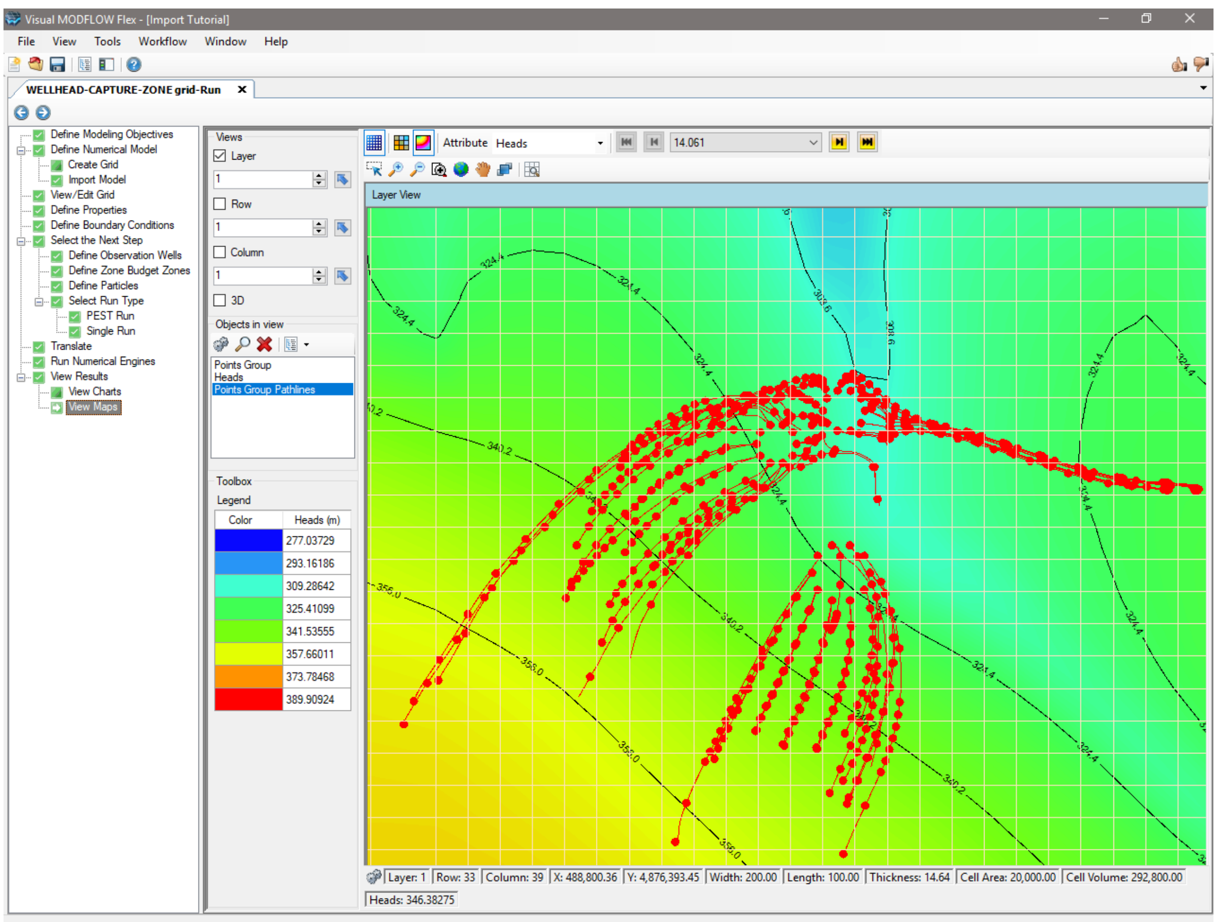
- Click the **[View Maps]** button.
- You will then see color shading of the calculated heads, in layer view.



- 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.

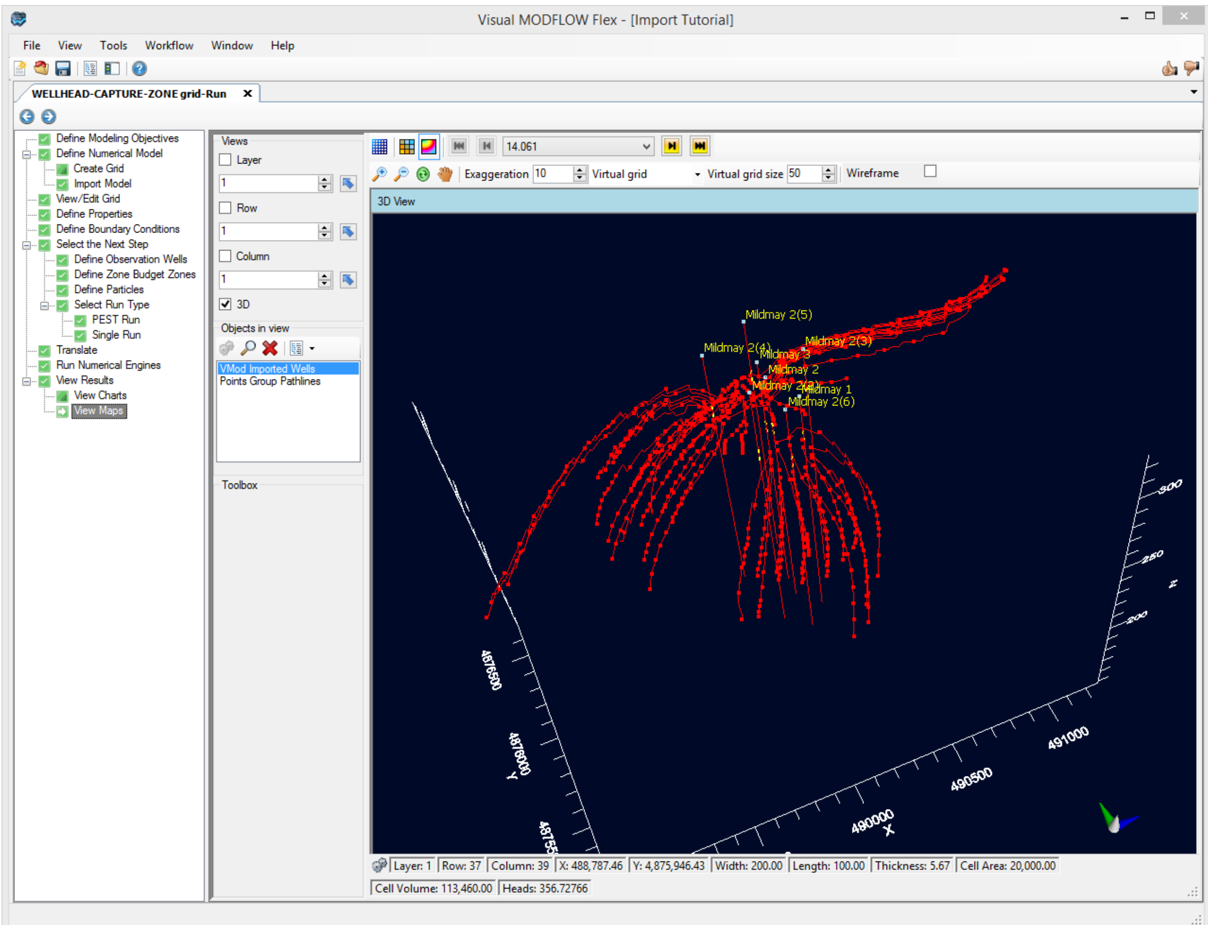
- If you ran MODPATH, you will see Pathlines appear as a new node in the 'Model Explorer' tree under Output; add a check box beside the **Backward Pathlines (i.e. [Points Group1 Pathlines])** to display these in the active 2D/3D Viewers. You may need to zoom into the middle of the layer view to see the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

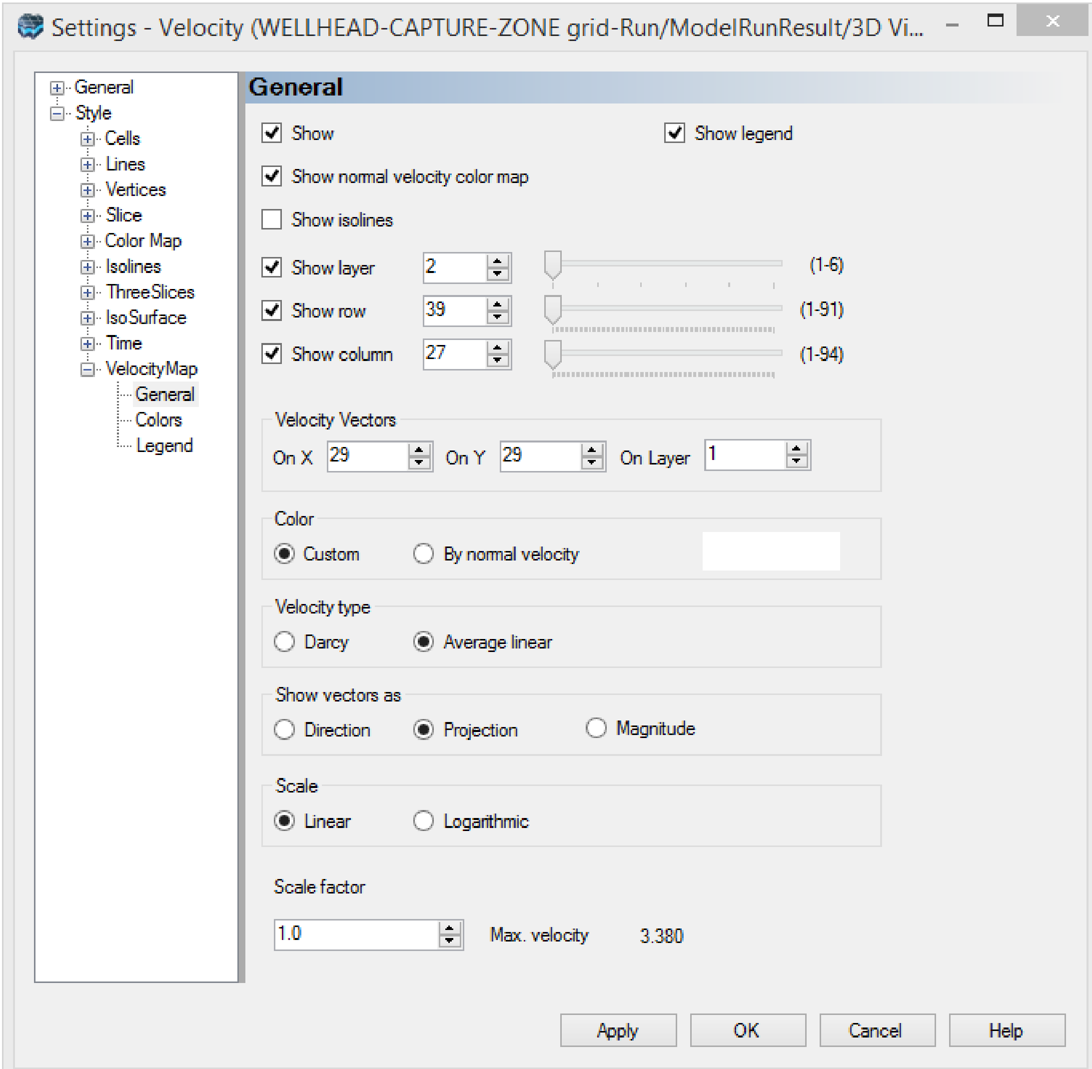
- Turn on the check box beside 3D View, in the available views
- Turn off the check box beside Layer View
- From the Model Explorer, turn off the check box beside Heads
- From the Data explorer (raw data), turn on the check box beside "VMod Imported Wells"
- Take a moment to rotate your 3D view, and 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:

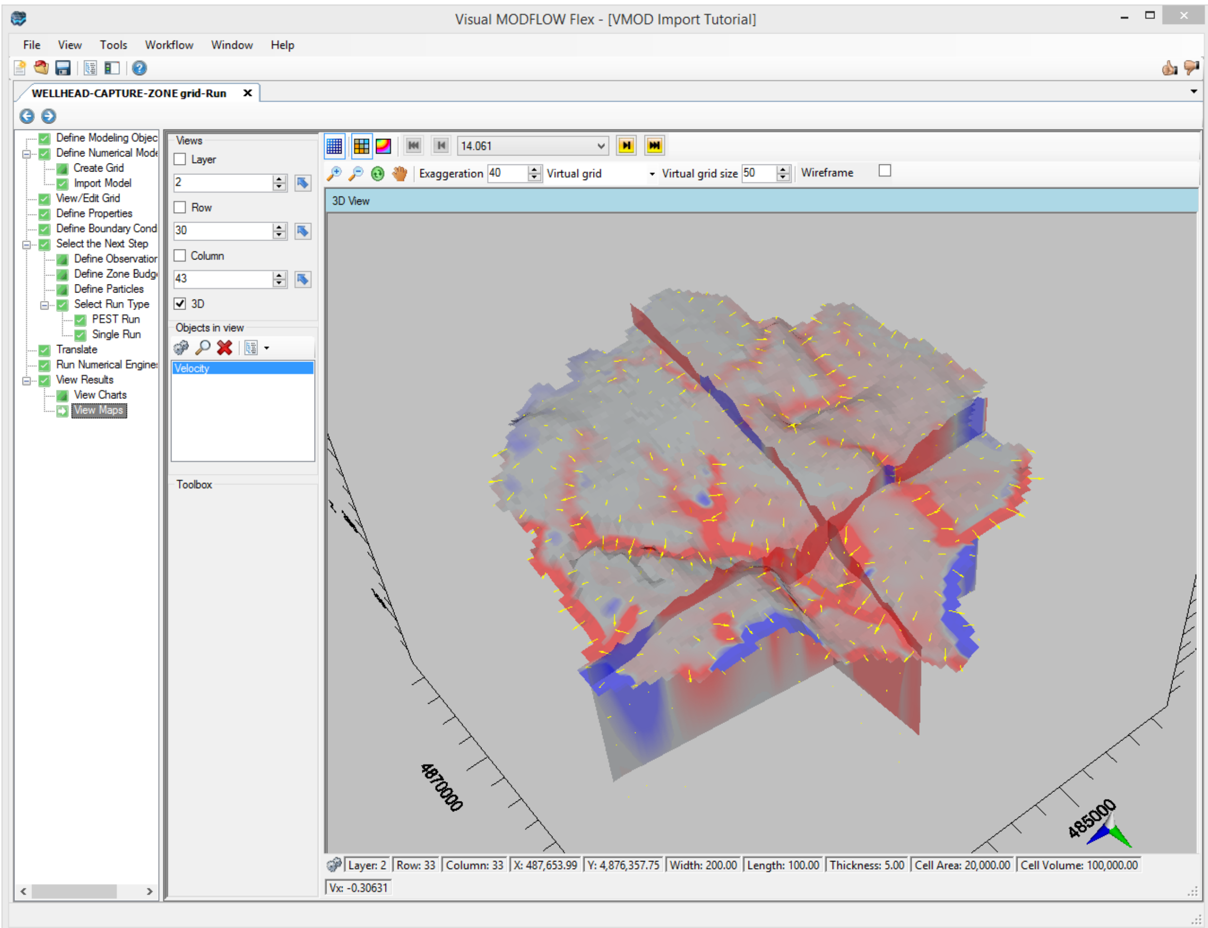


Velocity vectors may be displayed using average linear or Darcy velocities. In-plane velocity vectors can be displayed using the full magnitude of the vector (i.e. the out-of-plane velocity component will be included), as a projection (i.e. only the in-plane velocity component is displayed), or simply 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.

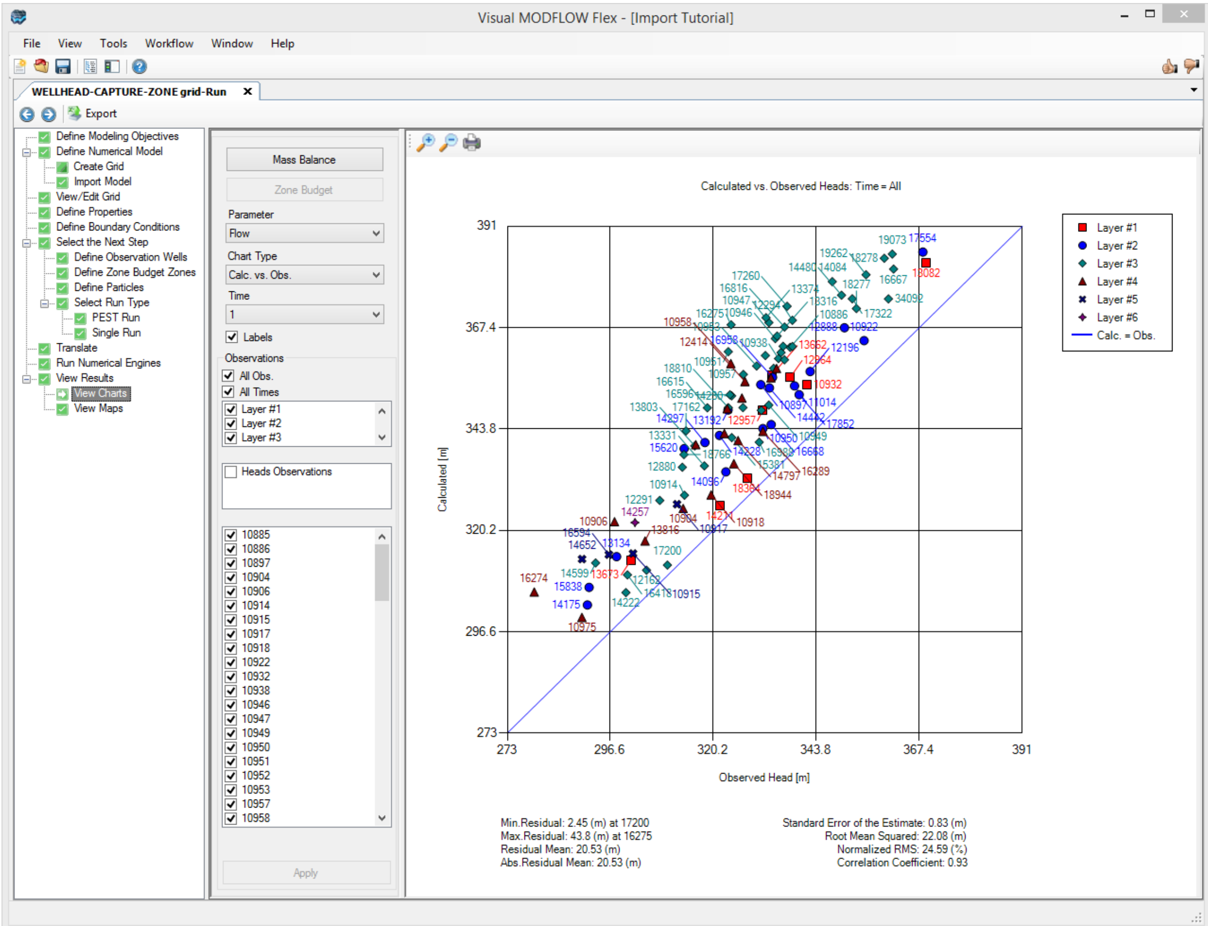
Update the velocity map settings as follows:

- 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.

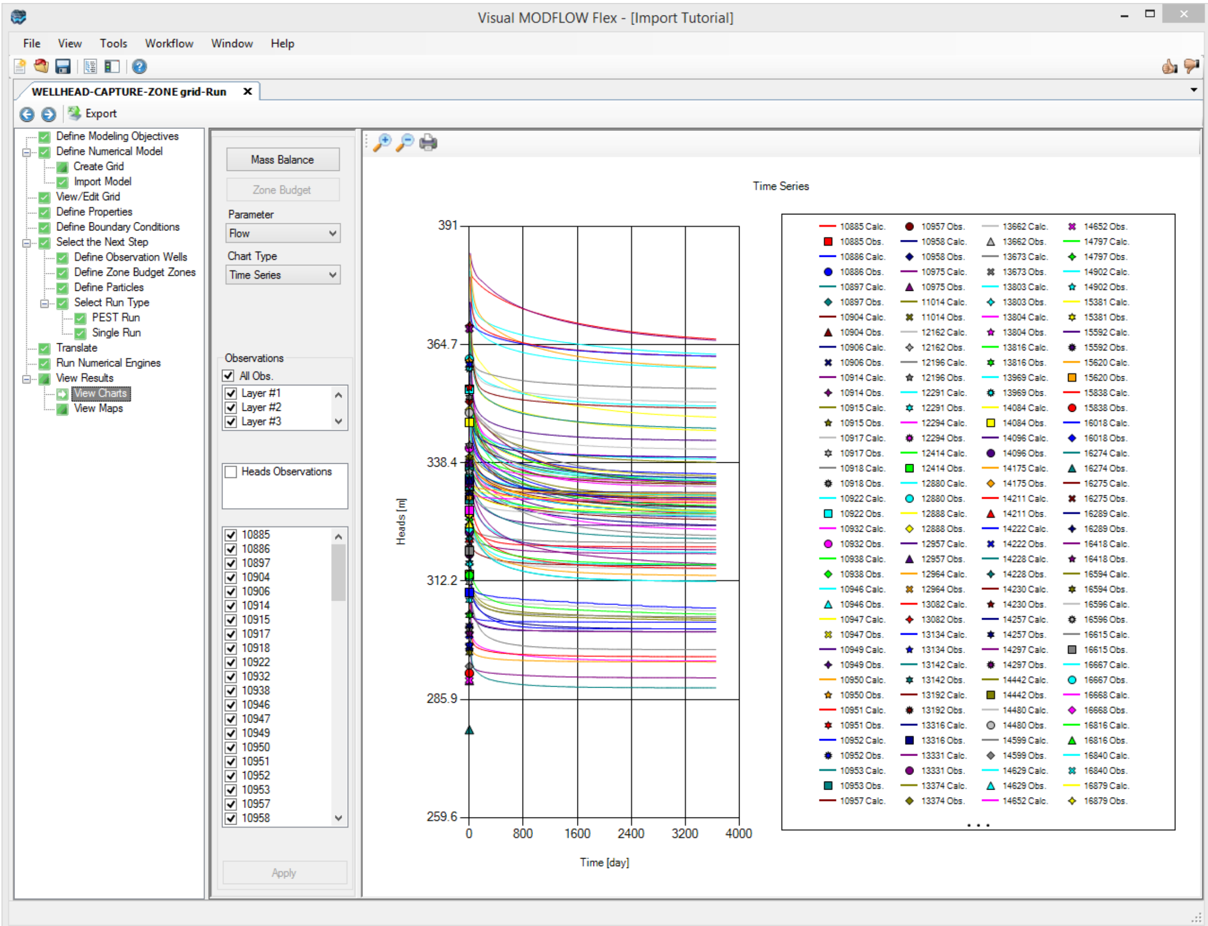


View Charts

- 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:



*******This concludes the 'Importing Visual MODFLOW/MODFLOW Models' tutorial.*******