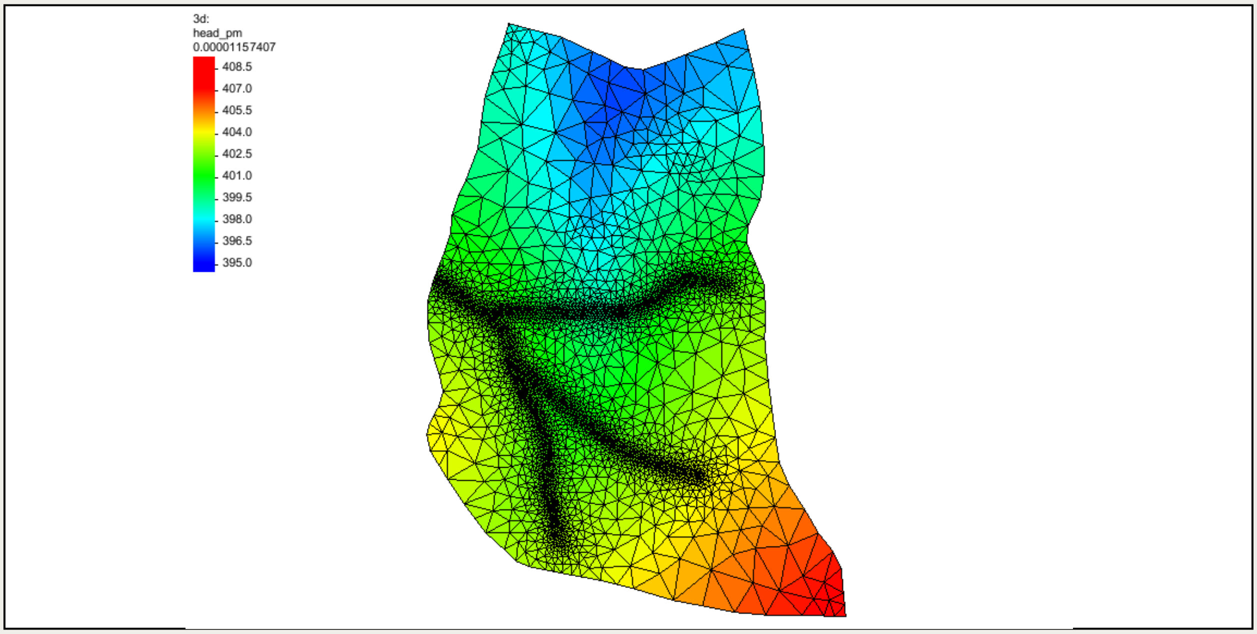
GMS 10.8 Tutorial

HydroGeoShere – Post-Processing

Review a HydroGeoSphere model in GMS



Objectives

This tutorial demonstrates post-processing options for a HydroGeoSphere model in GMS.

Prerequisite Tutorials

- Getting Started
- HydroGeoSphere – Porous Media Domain

Required Components

- GMS Core
- HydroGeoSphere Model & Interface

Time

- 25–45 minutes

1 Introduction

This tutorial demonstrates post-processing options for a HydroGeoSphere (HGS) model in GMS. It is based off of *the Intro to HydroGeoSphere* tutorial from Aquanty. Refer to that tutorial and the *HydroGeoSphere Reference Manual* as needed.

In the *Intro to HydroGeoSphere* tutorial, the "R5 Catchment" model is developed. From that tutorial:

> The model is quite simple compared to many 'real world' models that you may build. But it contains sufficient complexity that it is perfectly suited to demonstrate the overall HydroGeoSphere workflow. The intention of this tutorial is not to perfectly recreate the catchment, but to teach the process of creating a small watershed model in HGS. In fact, several of the inputs for the R5 model are completely arbitrary (e.g. the delineation of soil types and associated hydraulic properties, boundary condition values, etc.).
>
> The R5 catchment covers a small (approximately 0.1 km2) rangeland catchment near Oklahoma City (Figure 3). The defined model includes basic flow in the porous medium and surface/overland flow domains, and includes evapotranspiration.[1]

[1] HydroGeoSphere Intro Tutorial, pg 13

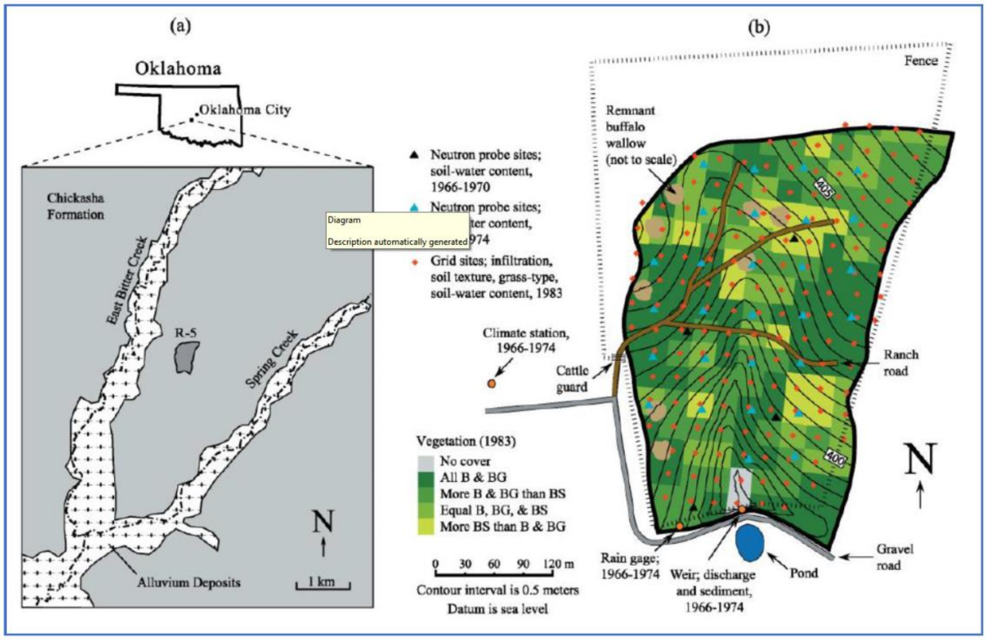


Figure 1 HydroGeoSphere Intro Tutorial, figure 3, pg 14

2 Basic Workflow

From the *Intro to HydroGeoSphere* tutorial, the basic HGS workflow consists of 5 steps:

1. Prepare data inputs (e.g. various GIS files, 2D grid/mesh files, etc.)
2. Configure an input data file (the GROK file) for the model pre-processor…
3. Use executables (*.exe) to generate the model input files and solve the simulation…
   a. Grok.exe – compiles the GROK file, generates input files for the numerical simulator
   b. Phgs.exe – reads model input files, performs the numerical simulation (i.e. it 'solves' the model)
   c. Hsplot.exe OR hgs2vtu.exe – reformats model outputs for visualization
4. Visualize, analyze and interpret model results…
5. Iterate on the model to add complexity, perform scenario analysis, calibrate, etc…[2]

Note that a GROK file is prepared and then the grok.exe runs. During the run, grok.exe reads the GROK file and prepares the actual model input files. Only then can the model (phgs.exe) be run.

The HydroGeoSphere workflow is followed pretty closely when using GMS. The basic steps to build an HGS model in GMS, and how these steps relate to the HydroGeoSphere workflow, are as follows:

1. Create the conceptual model (HGS step 1)

[2] HydroGeoSphere Intro Tutorial, pg 13

2. Create a 3D UGrid for the porous media domain (HGS step 1)
3. Create a 2D UGrid for the 2D domains: surface flow, ET (HGS step 1)
4. Save the simulation, which creates the GROK file (HGS step 2)
5. Run grok.exe on the GROK file to prepare input files for phgs.exe (HGS step 3a)
6. Run phgs.exe (HGS step 3b)
7. Read the solution and visualize the results (HGS step 4)
8. Iterate on the model to add complexity, perform scenario analysis, calibrate, etc… (HGS step 5, not covered in this tutorial)

3 Getting Started

Do as follows to get started:

1. If necessary, launch GMS.
2. If GMS is already running, select *File* | **New** command to ensure that the program settings are restored to their default state.
3. Click **Open** to bring up the *Open* dialog.
4. Select "Project Files (*.gpr)" from the *Files of type* drop-down.
5. Browse to the *hgs-post* folder and select "start.gpr".
6. Click **Open** to import the project and exit the *Open* dialog.

The project should be visible in the Graphics Window (Figure 2). The project contains a HydroGeoSphere simulation that has already been set up to include a 3D UGrid. Boundary conditions, materials, and the model control parameters have been set.

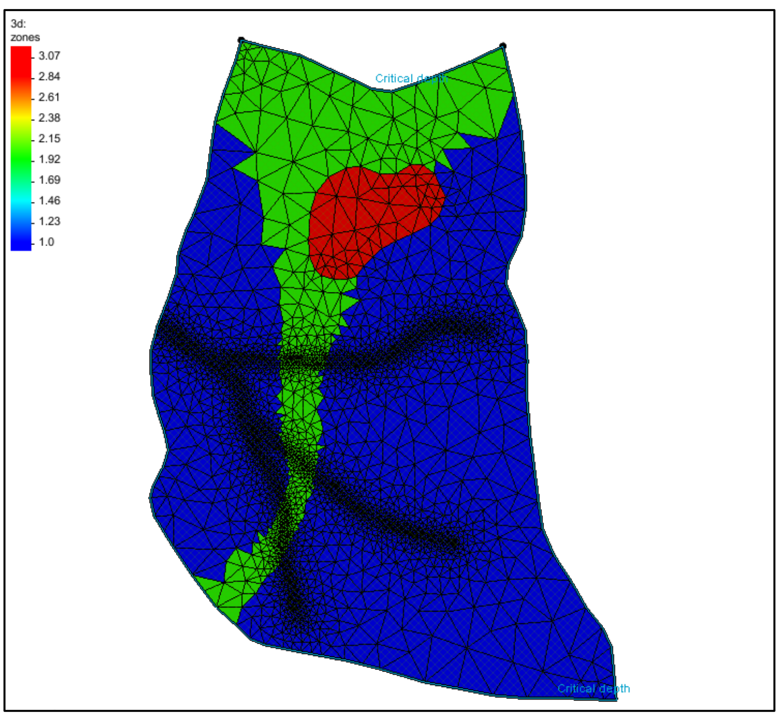


Figure 2 The initial project

Before continuing, it is recommended to save the project with a new name.

1. Select *File* | **Save As…** to bring up the *Save As* dialog.
2. Browse to the directory for this tutorial.
3. Select "Project Files (*.gpr)" from the *Save as type* drop-down.
4. Enter "hgs_post.gpr" as the *File name*.
5. Click **Save** to save the project and close the *Save As* dialog.

4 Model Output

GMS allows recording both observation and hydrograph data from the HydroGeoSphere model run. Doing this requires creating map coverages that can then be added to the simulation.

4.1 Observations

Create an observation coverage by doing the following:

1. Turn off the " 3d" UGrid.
2. Right-click in the Project Explorer and select *New* | **Coverage…** to generate a new coverage
3. Right-click on " new coverage" and select **Rename**.
4. Enter "observations" for the new name.
5. Right-click on the " observations" coverage and select *Coverage Type* | *HydroGeoSphere* | **Observations**.
6. Using the **Create Point** tool, create a point anywhere in the model area.
7. Keeping the new point selected, enter following coordinates:
    - *X* = 360
    - *Y* = 295
    - *Z* = 398
8. Using the **Select Points\Nodes** tool, double-click the point to open the *Point Properties* dialog.
9. In the list on the left, turn on *Observation point*.
10. For the *Name* enter "point1".

The Interpolate option causes computed values from nearby points to be interpolated to the observation point (corresponding to the "Make interpolated observation point" command). If off, computed values from the nearest point are used (corresponding to the "Make observation point" command). For this example, leave it off.

11. Click **OK** to close the *Point Properties* dialog.
12. Drag the " observations" coverage below the " R5" simulation.

4.2 Hydrographs

Create an hydrograph coverage by doing the following:

1. Right-click on the " surface flow BCs" coverage and select **Duplicate**.

2. Right-click on "Copy of surface flow BCs" and select **Rename**.
3. Enter "hydrographs" for the new name.
4. Right-click on the "hydrographs" coverage and select *Coverage Type* | *HydroGeoSphere* | **Hydrographs.**
5. Using the **Select Arc** tool, double-click the top arc to open the *Arc Properties* dialog.
6. In the list on the left, turn on *Hydrograph*.
7. Set the *Name* to "Outlet".
8. Turn on the *Map to grid boundary* option.
9. Click **OK** to close the *Arc Properties* dialog.
10. Drag the "hydrographs" coverage below the "R5" simulation.

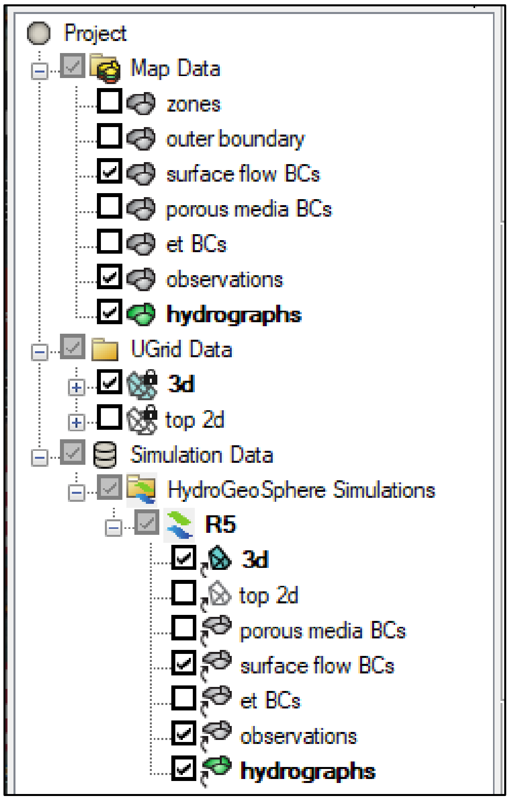


Figure 3 The HydroGeoSphere simulation in the Project Explorer

5 Save the Project and Simulation

Before continuing, complete the following:

1. **Save** the project.
2. Right-click on the "R5" simulation and select **Save Simulation** to open the *Saving HydroGeoSphere simulation* dialog.
3. When finished, click **OK** to close the *Saving HydrogGeoSphere Simulation* dialog.

Saving the simulation creates the GROK file and the other files that it refers to. To view these files:

4. Right-click on the " R5" simulation and select **Open Containing Folder** to open a Windows File Explorer.

The opened file explorer shows the directory where GMS saved the model files.

6 Run Grok

1. Right-click the " R5" simulation and select the **Run grok** command to open a run dialog.

The "Running grok.exe" dialog appears. The output from grok.exe is displayed. It should only take a few seconds for grok.exe to finish. When it is done, there may be some warnings at the bottom of the output but there should not be any errors.

2. When the process finishes, click **OK**.

7 Run Phgs

Now that the input files have been prepared by running grok.exe, the model is ready to be run.

1. Right-click the " R5" simulation and select the **Run phgs** command.

The "Run phgs.exe" dialog appears. The progress dialog shows a plot of the residual error versus time, and the output from phgs.exe. It will take a few minutes for phgs.exe to finish.

2. When the process finishes, click **OK**.

8 Examine the Solution

8.1 Read the Solution

1. Right-click the " R5" simulation and select **Read Solution** to start the *Read Solution* progress dialog.

When GMS is done reading the solution, the *Read Solution* progress dialog should disappear automatically if there are no errors.

A new Solution folder was added under the R5 simulation, with subfolders "pm", and "olf". These contain links to datasets that were added under "R5 (HydroGeoSphere)" folders under the "3d" and "top 2d" UGrids. Let's look at some of the datasets.

8.2 3D Datasets

1. Turn off all " Map Data".
2. Turn on the " 3d" UGrid.
3. Under the " 3d" UGrid, expand the " R5 (HydroGeoSphere)" and " pm" folders, select the " head_pm" dataset.

4. In the *Time Step* window below the Project Explorer, select the **Options** button to open the *Time Settings* dialog.
5. Change the *Relative* option to "seconds".
6. Click **OK** to close the *Time Settings* dialog.
7. Select different time steps and see that the contours change a little with time.

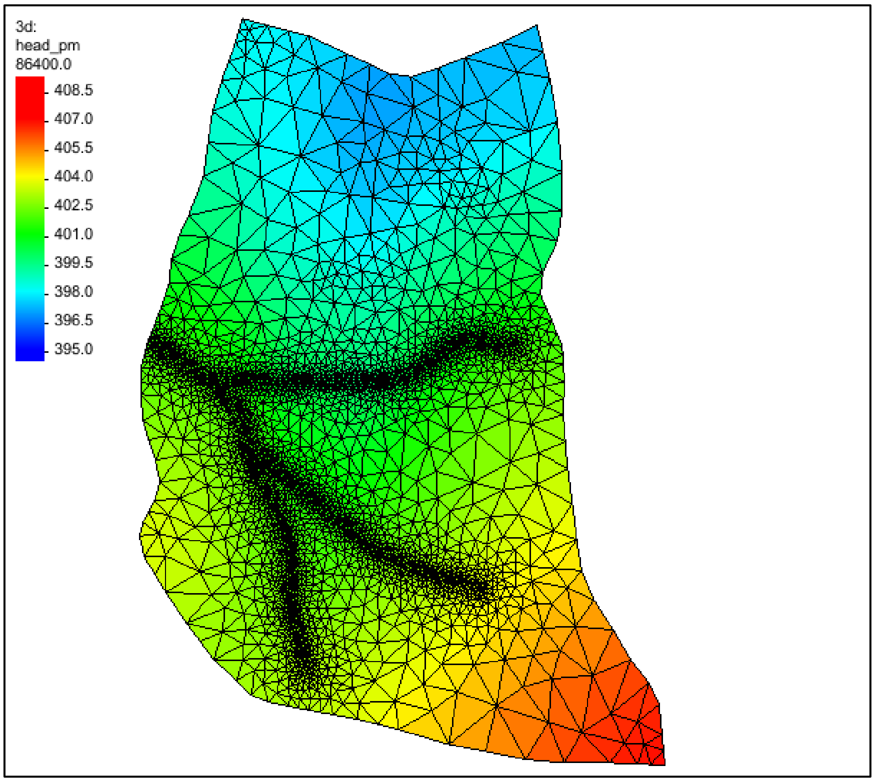


Figure 4 3D "head_pm" dataset

8.3 3D Datasets Vectors

1. Under the " pm" folder, select the " q_pm" dataset.
2. Click the **Display Options** macro to open the *Display Options* dialog.
3. Turn off *Cell edges* and turn on *Vectors*.
4. Click **OK** to close the *Display Options* dialog.
5. Select different time steps and see that the vectors change a little with time.

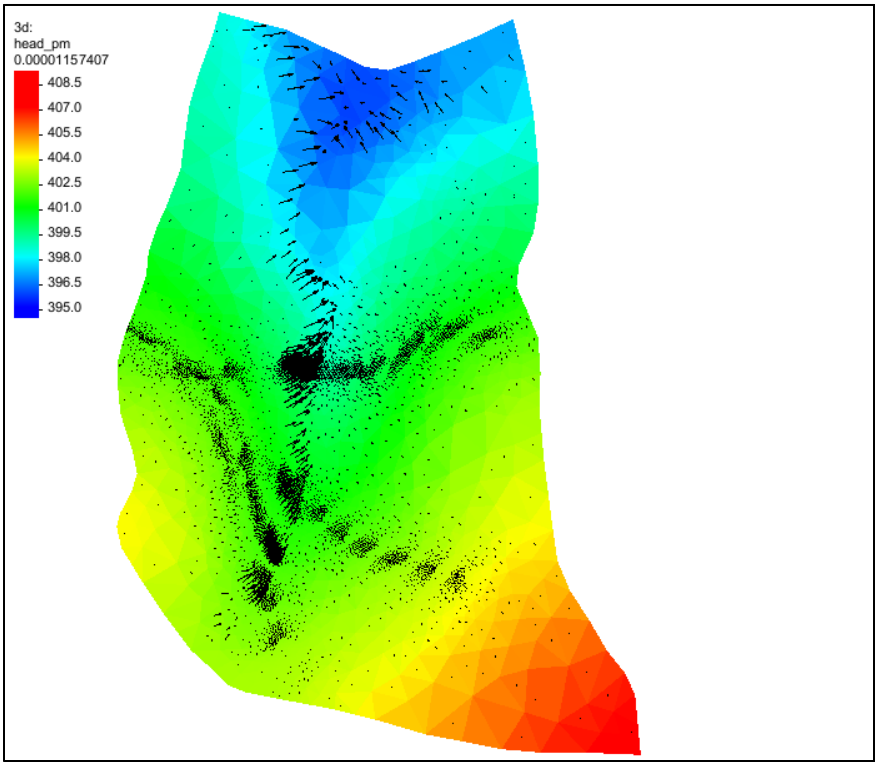


Figure 5 3D "q_pm" vectors

8.4 2D Datasets

1. Turn off the " 3d" UGrid and turn on and select the " top 2d" UGrid.
2. Under the " top 2d" UGrid, expand the " R5 (HydroGeoSphere)" and " olf" and " BCs" folders and select the " CritDepth_outlet-Flux (nodal)" dataset.

All the datasets in the " BCs"folder were created by boundary conditions. The " CritDepth_outlet-Flux (nodal)" dataset was created by the critical depth boundary condition on the top arc. The dataset has meaningful values at the points where the boundary condition arc is defined and null values at all other points.

3. Click the **Display Options** macro to open the *Display Options* dialog.
4. Turn off *Face contours* and *Vectors* then turn on *Cell edges* and *Point contours*.
5. Click **OK** to close the *Display Options* dialog.
6. Select the last time step and see that the point contours change.

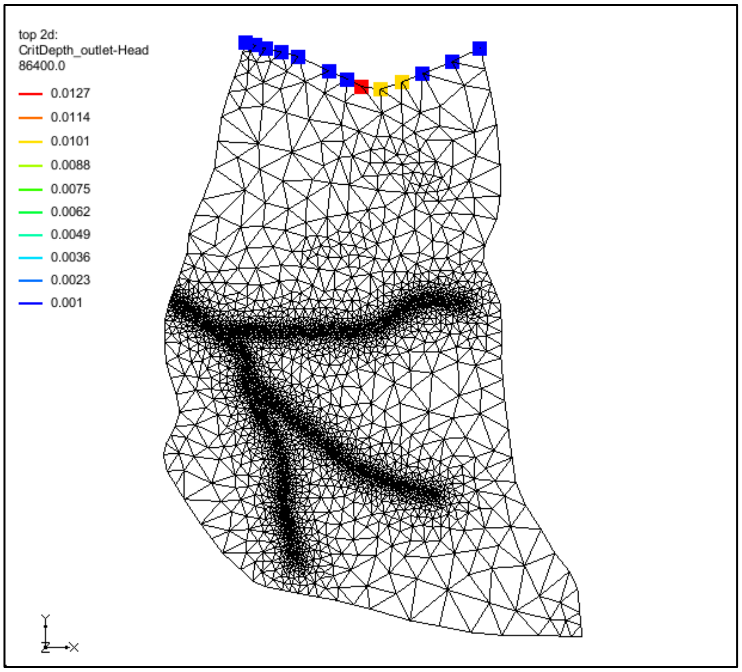


Figure 6 2D " CritDepth_outlet-Flux (nodal)" dataset

8.5 Solution Plots

In addition to datasets on the UGrids, GMS also read in a number of plots when the solution was read.

1. Right-click the " R5" simulation and select the **Solution Plots** command to open the *Solution Plots* dialog.

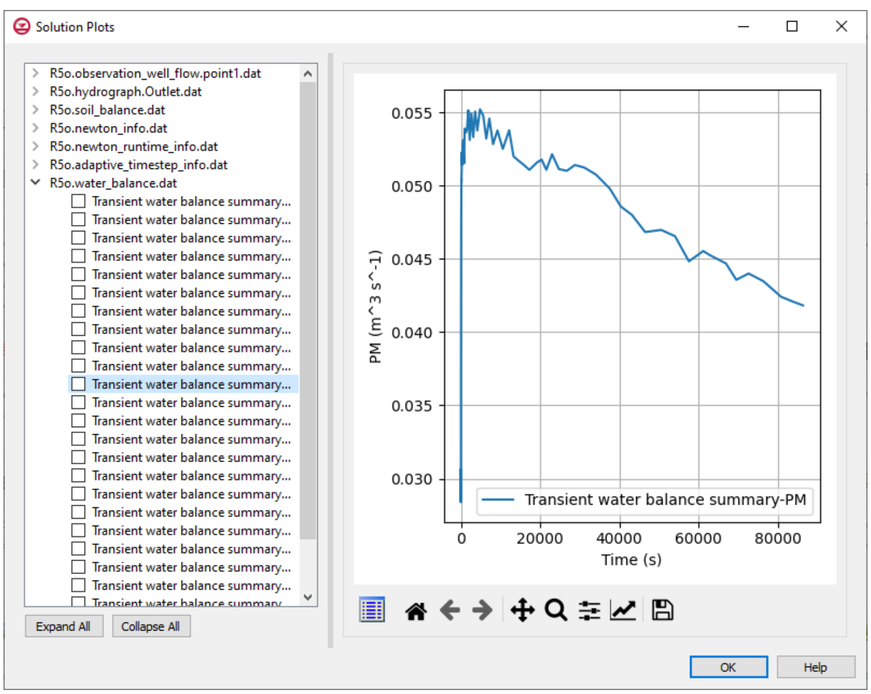


Figure 7 HydroGeoSphere solution plots

The window on the left of the dialog lists the plots that can be displayed. They are grouped by the file that they came from. The currently selected item is displayed in the plot window on the right. Multiple plots can be displayed simultaneously by toggling them on in this list. Notice there are plots for the boundary conditions ("Transient water balance summary-Well" etc), the observation point (under "R5o.observation_well_flow.point1.dat") and the hydrograph (under "R5o.hydrograph.Outlet.dat"). The water balance file plots are displayed by default.

2. Expand other files and click on different plots to see the plot data generated by HydroGeoSphere.
3. Click on the **View Data** button to open the *Plot Data* dialog to see the plot data values.

9 Conclusion

This concludes the "HydroGeoSphere Post-Processing" tutorial.

