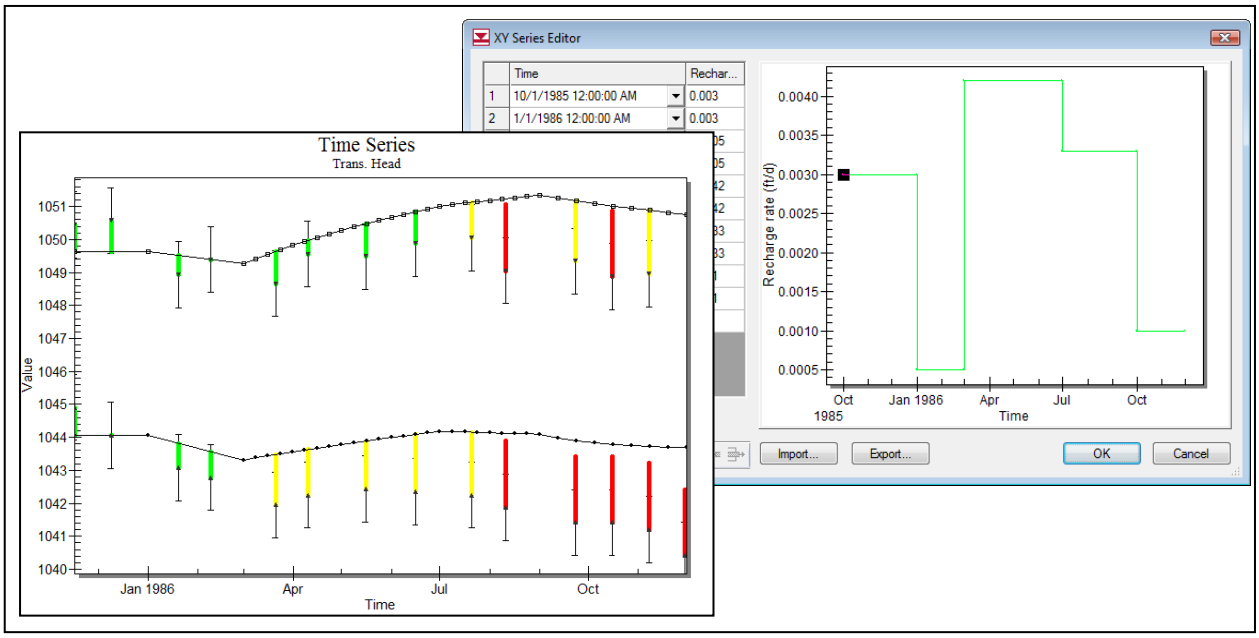
GMS 10.0 Tutorial

MODFLOW – PEST Transient Pump Test Calibration

Tools for calibrating transient MODFLOW models



Objectives

Learn how to setup a transient simulation, import transient observation data, and use PEST to calibrate the model.

Prerequisite Tutorials

- MODFLOW – PEST Pilot Points

Required Components

- Grid Module
- Map Module
- MODFLOW

Time

- 30-45 minutes

AQUAVEO™

1 Introduction **2**
1.1 Outline 2
2 Description of Problem **2**
3 Getting Started **4**
4 Reading in the Project **4**
5 Save the Project With a New Name **5**
6 Setting up the Transient MODFLOW Model **5**
6.1 Entering MODFLOW Stress Period Data 6
6.2 Initial Conditions 7
7 Entering Pumping Data **7**
8 Importing Transient Observation Data **8**
8.1 Adjusting the Coverage Set Up 8
8.2 Importing Transient Data Text File 8
9 Saving and Running MODFLOW **10**
10 Creating a Time Series Plot **11**
11 Running PEST **11**
12 Pilot Points **13**
12.1 Using Pilot Points with the Parameters 14
13 Running PEST **14**
14 Conclusion **16**

1 Introduction

Pump test data is a common type of transient data available to groundwater model developers. In this tutorial, the user will take an existing steady state MODFLOW model and update the model to simulate a pump test.

1.1 Outline

Here are the steps to this tutorial:

1. Open a MODFLOW model and solution.
2. Set up MODFLOW stress periods.
3. Import transient observation data and create plots.
4. Run PEST to calibrate the transient model.

2 Description of Problem

The model in this tutorial is the same model used in the "MODFLOW – Generating Data From Solids" tutorial. Figure 1, below, shows the study area in plan view. The production well near the center of the study area was pumping during the pump test. The other two production wells were not pumping during the test. A few head measurements were taken at these wells during the test. One monitoring well was sampled frequently during the pump test. Other monitoring wells underwent less frequent measurements.

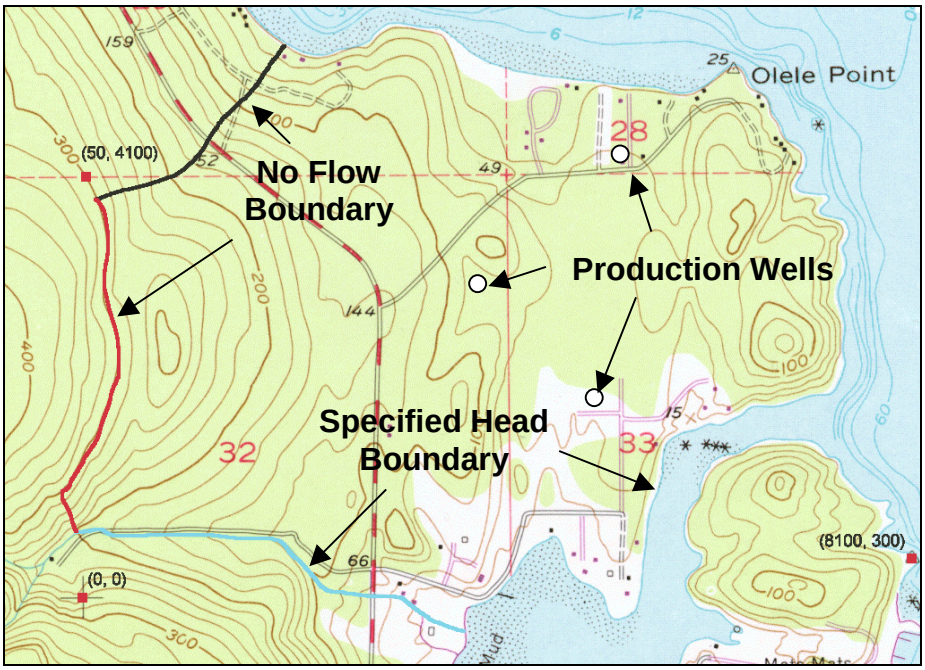


Figure 1 Study area for ground water model

The figure below shows a cross section through the study area. There are two main aquifers in the area. The lower-confined aquifer (red) is overlain by an upper unconfined aquifer (green). The lower aquifer is where the pumping well and main observation well are located. In some areas, there is an aquitard (yellow) that overlays portions of the lower aquifer.

Figure 2 Cross section through the study area

The user will import transient observation data at multiple observation wells. The model has already been parameterized into different zones of hydraulic conductivity (HK), specific yield (SY), and specific storage (SS) for the upper and lower aquifers; there is also a parameter for estimating recharge. The user will run the model with the current parameter values to see how well the model matches the pump test. Then the user will

have PEST optimize the parameter values. Finally, the user will use pilot points with the parameters to see if it is possible to improve the match between the simulated and field-observed values.

3 Getting Started

Do the following to get started:

1. If necessary, launch GMS.
2. If GMS is already running, select the *File* | **New** command to ensure that the program settings are restored to their default state.

4 Reading in the Project

First, read in the project:

1. Select the **Open** button.
2. Locate and open the *Tutorials\MODFLOW\trans_pest_pumptest* directory.
3. Select the file entitled "start.gpr."
4. Click **Open.**

The user should see a MODFLOW model with a solution and a set of map coverages similar to the next figure.

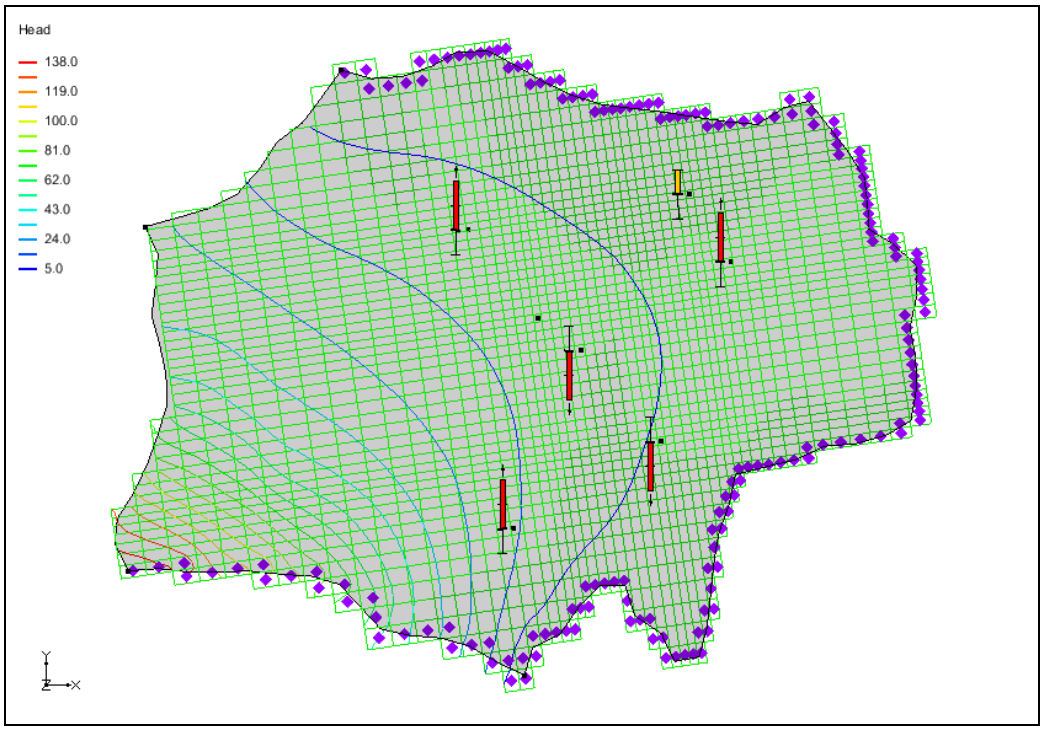


Figure 3 Steady state MODFLOW model

5 Save the Project With a New Name

Before continuing, save the project with a new name.

1. Select the *File* | **Save As** command.
2. Change the project name to **"**pumptest.gpr."
3. Click **Save.**

It's a good idea to save the work periodically.

6 Setting up the Transient MODFLOW Model

First, the user is going to set up the transient MODFLOW model. The pump test occurred over a two-week period. Data was collected at the main observation well during the first three days while the well was pumping and then more sporadically over the next ten days while the well was off. The user will want to set up the MODFLOW model to have at least two stress periods: one stress period with the well on for 3 days and one stress period with the well off for 10 days.

1. Select the *MODFLOW* | **Global Options** command to open the *MODFLOW Global/Basic Package* dialog.

2. On the right side of the dialog, change the *Model type* to *Transient*.

3. Click on the **Stress Periods** button.

6.1 Entering MODFLOW Stress Period Data

This brings up the *Stress Periods* dialog. Here, it is possible to set the number of stress periods, how long each period will be, and the number of time steps in each period.

1. Increase the *Number of stress periods* to "3."

2. Change the *Start* of the second stress period to "0.0" in the spreadsheet. The *Length* of the first stress period will be updated automatically to 0.0.

3. Change the *Start* of the third stress period to "3.0." The *Length* of the second stress period will be updated automatically to 3.0.

4. Change the *End* of the third stress period to "13.0." The *Length* of the third stress period will be updated automatically to 10.0.

5. Make sure that the *Num. Time Steps* is "10" for stress periods 2 and 3.

6. Set the *Multiplier* to "1.5" for stress periods 2 and 3. This increases the number of time steps at the beginning of the stress period so that the user can see the effect of the change in stresses more accurately.

When the user is finished, the dialog should look like the figure below.

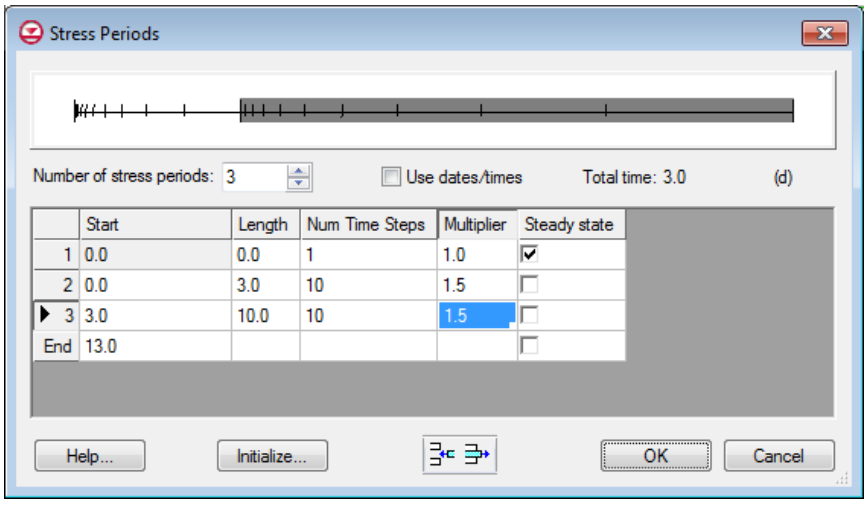


Figure 4 Stress periods dialog

7. Select **OK** to exit the dialog.

8. Select **OK** at the prompt. The first stress period should have a length of zero because it is a steady state stress period.

6.2 Initial Conditions

With a transient model, it is important to have an initial condition that is consistent with the model inputs. If the initial condition is not consistent with model parameters, then the model response in the early time steps will reflect not only the model stresses but also the adjustment of computed head values to offset the lack of correspondence between the model inputs and the initial head values.[1] In this case, the user has set the first stress period to be steady state. This will prevent any lack of correspondence between model inputs and the initial heads for the transient stress periods. This is important when calibrating a transient model. When the user allows a tool like PEST to change the model inputs, it is necessary to make sure that the initial heads correspond to the new inputs that PEST has chosen. The easiest way to do this is to have the first stress period of the model be steady state.

1. Select **OK** again to exit the *MODFLOW Global/Basic Package* dialog.

7 Entering Pumping Data

It is now necessary to update the well data so that the well will have the appropriate pumping rate for the first stress period and 0.0 for the second stress period. The user will edit the well data in the WEL Package.

1. Select the *MODFLOW* | *Optional Packages* | **WEL - Well** menu command to open the *MODFLOW Well Package* dialog.

This dialog contains a spreadsheet listing all of the pumping wells in the model. Currently, all of the wells have a flow rate of 0.0. The user is going to change the flow rate for the second well, named "w-15," to -130,000.0 during the second stress period.

2. Near the top of the dialog, change the *Stress period* to "2."
3. Uncheck the *Use previous* check box next to the *Stress period* field.
4. Enter "-130000.0" for the *Q (flow)* for the second well listed in the spreadsheet.
5. Near the top of the dialog, change the *Stress period* to "3."
6. Uncheck the *Use previous* check box next to the *Stress periods* field and make sure the *Q (flow)* is "0.0" for the second well.
7. Select **OK** to exit the dialog.

1 Mary P. Anderson and William W. Woessner, *Applied Ground Water Modeling* (Waltham, Massachusetts: Academic Press, 1992).

8 Importing Transient Observation Data

Now the user will import the transient field-measured head values. This type of data can be imported using the *Text Import Wizard*. The *Text Import Wizard* can import transient data in multiple formats, including date/time. In this example problem, the user is not using date/time-formatted transient data; instead, the user is using relative time where the beginning of the simulation is time 0.0 and the units are days. Below, the user can see the format for this kind of data.

```
Name            Time        Head
mw - 3          0.0         10.25
mw - 3          0.5         10.3
w-38            15.0        4.6
…
```

8.1 Adjusting the Coverage Set Up

Before importing the transient observation data, the user needs to make sure that the coverage with the observation points is set up correctly.

1. Expand the "MODFLOW" conceptual model under the "Map Data" folder.
2. Double-click on the "Obs" coverage in the Project Explorer.
3. On the right side of the dialog under *Observation Points*, select the check box next to *Trans. Head.*
4. Select **OK** to exit the dialog.

8.2 Importing Transient Data Text File

It is now possible to import the transient observation data.

1. Select the **Open** button.
2. Navigate to the *Tutorials/MODFLOW/trans_pest_pumptest* directory.
3. In the *Open* dialog, change the *Files of type* selection to "Text Files (*.txt)."
4. Select the file named "obs_wells_trans.csv."
5. Click **Open**.

The user should now see the *Text Import Wizard*. It should look similar to the next figure. This wizard is used to import text data into a GMS project.

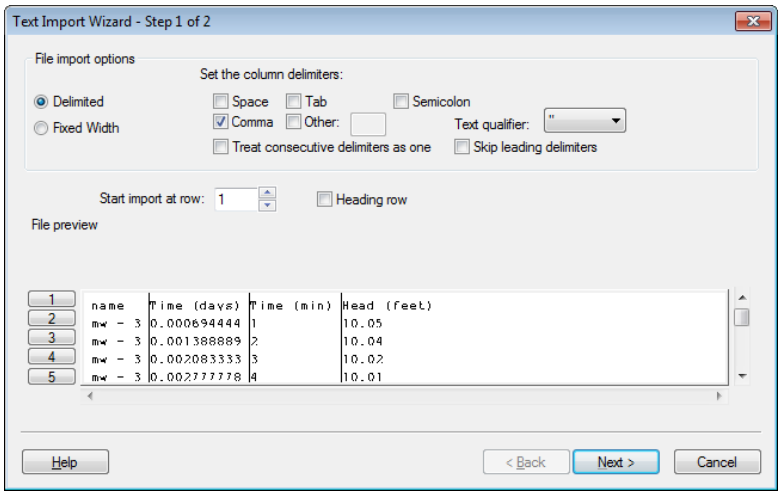


Figure 5 Text Import Wizard

6. Turn on the *Heading row* check box and click the *Next* button.

7. Near the top of the dialog, change the *GMS data type* to "Transient Observation data." The dialog should look like the next figure.

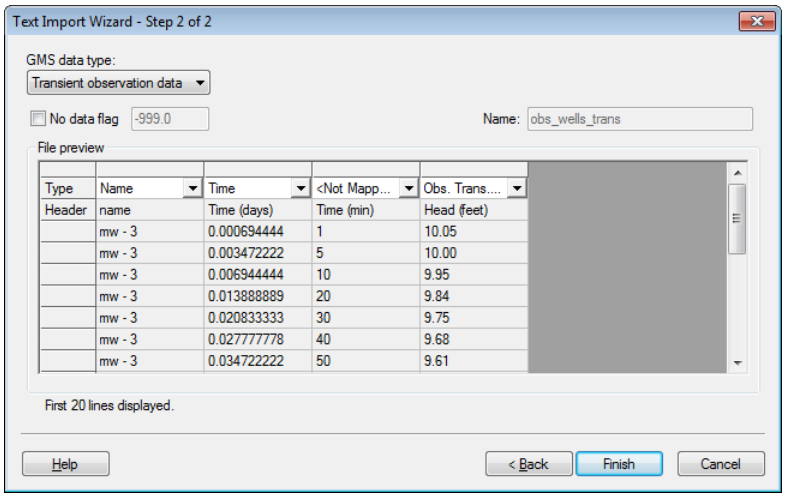


Figure 6 Step 2 of the Text Import Wizard

Notice that the Name, Time, and Head columns were automatically recognized by GMS. There are two time columns, one in units of days and one in units of minutes. The user will want to use the one in units of days and ignore the other one. The dialog is already set up in the desired way.

8. Click on the **Finish** button.

The user can see the imported time series data by double-clicking on any of the observation wells and selecting the button under the *Obs. Trans Head* column. The next figure shows the time series curve for observation well "mw - 3." This observation well is near the center of the model.

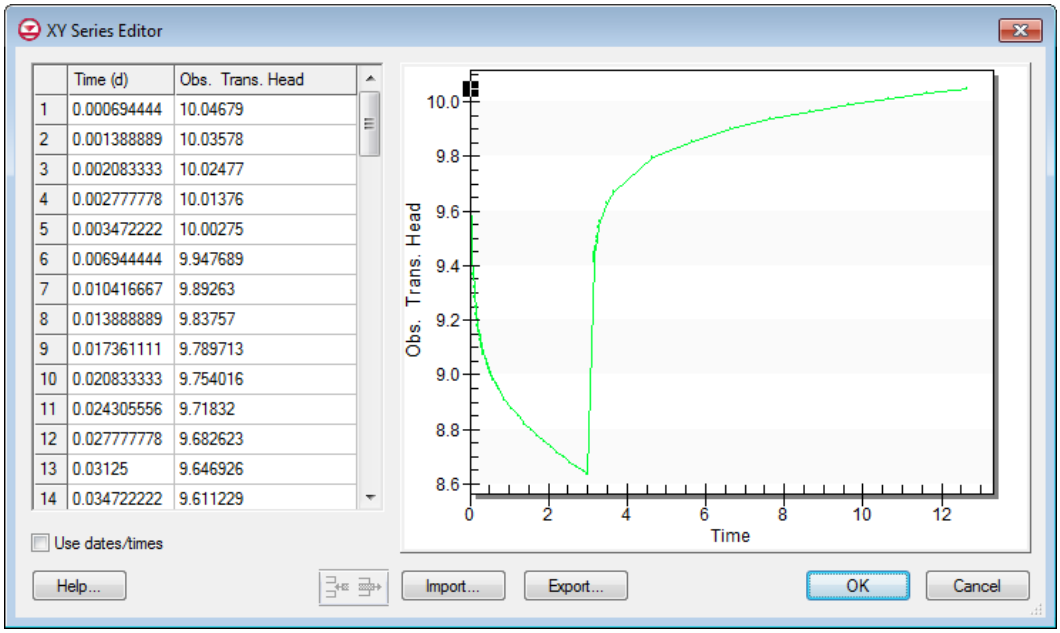


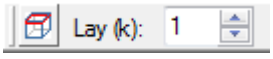
Figure 7 Time series data for well "mw - 3"

9 Saving and Running MODFLOW

The user is now ready to save the model and launch MODFLOW.

1. Select the **Save** button.
2. Select the **Run MODFLOW** button.
3. Once MODFLOW has finished, select the **Close** button to close the window and return to GMS.

The contours should change. The user is currently looking at the top layer of the grid. The pumping well and some of the observation wells are in layer 5.

4. Use the *Mini-grid Toolbar* 

near the top of the GMS window to change the current layer to layer **5**.
5. Expand the "pumptest (MODFLOW)" item in the Project Explorer.
6. Select the "Head" dataset.
7. Use the *Time Step* window to cycle through the different time steps of the solution to see the effect of the pump test.

10 Creating a Time Series Plot

One of the useful tools for working with transient calibration data is the time series plot. This plot allows us to see how well the simulated heads match the field measurements.

1. Select the **Plot Wizard** button.
2. In the *Plot Wizard* dialog, change the plot type to *Time Series*.
3. Select the **Next** button.
4. Select the check box next to point *mw - 3*.
5. Also, turn on the *Calibration Target* and *Observed Values* check boxes.
6. Select the **Finish** button.

The user should see a plot similar to the next figure. Currently, the model simulated heads are a little low at this particular well. The user will now use PEST to help calibrate the transient model. The user can also see other observation wells in the model by right-clicking on the plot and selecting the Plot Data command and selecting a different well in the dialog.

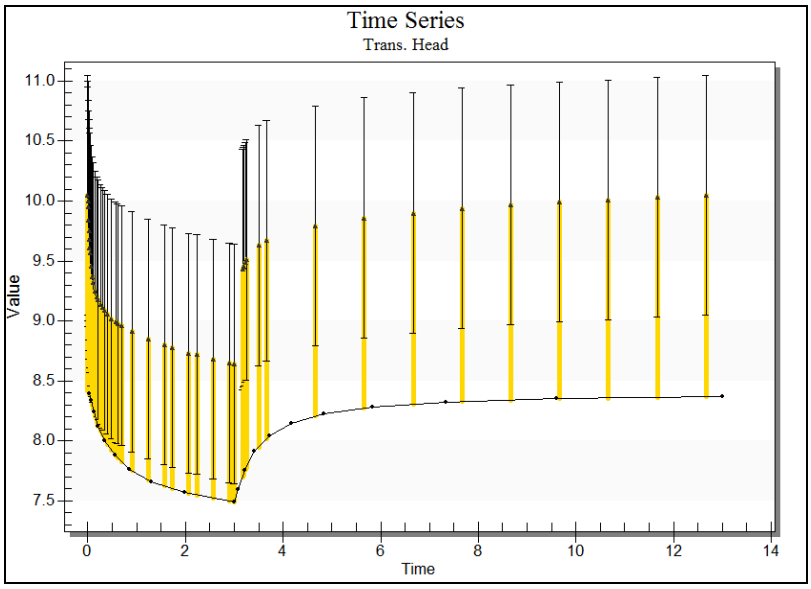


Figure 8 Time series plot

11 Running PEST

The user will change the MODFLOW simulation so that PEST is used to estimate the values of HK (hydraulic conductivity), SY (specific yield), and SS (specific storage) of the upper and lower aquifer. The user will also allow PEST to estimate the value of recharge (RCH).

1. Select the *MODFLOW* | **Global Options** command to open the *MODFLOW Global/Basic Package* dialog.

2. Under the *Run options*, select the *Parameter Estimation* option.

3. Select **OK** to exit the dialog.

At this point, the user would normally need to parameterize the model. Model parameterization is explained in the "MODFLOW – Automated Parameter Estimation" tutorial. This model has already been parameterized, so it is now possible to run PEST. (This model uses material zones to assign properties to the aquifers. The HK, SY, and SS parameter key values have been entered in the *Material Properties* dialog. If the user is interested in seeing the materials properties select the *Edit* | **Material Properties** command.) Now the user is going to use Parallel PEST to optimize this model.[2]

4. Select the *MODFLOW* | **Parameter Estimation** command to open the *PEST* dialog.

5. Make sure that the *Use Parallel PEST* and *Use SVD* options are checked.

6. Select **OK** to exit the dialog.

It is now possible to run Parallel PEST.

7. Select the *File* | **Save As** menu command.

8. Change the project name to "pumptest_pest.gpr."

9. Select **Save.**

10. Select the **Run MODFLOW** button.

Depending on the speed of the user's computer, Parallel PEST will take about 2 to 10 minutes to run this problem. When finished, the user should see that PEST completed 2 iterations and had a final model error of about 697.

11. Select the **Close** button.

The contours and time series plot should update. The time series curve for the observation well should look like the next figure. The user has made a minor improvement in the match with the field-observed values.

2 This next step can be completed without Parallel PEST but it will take significantly longer to complete.

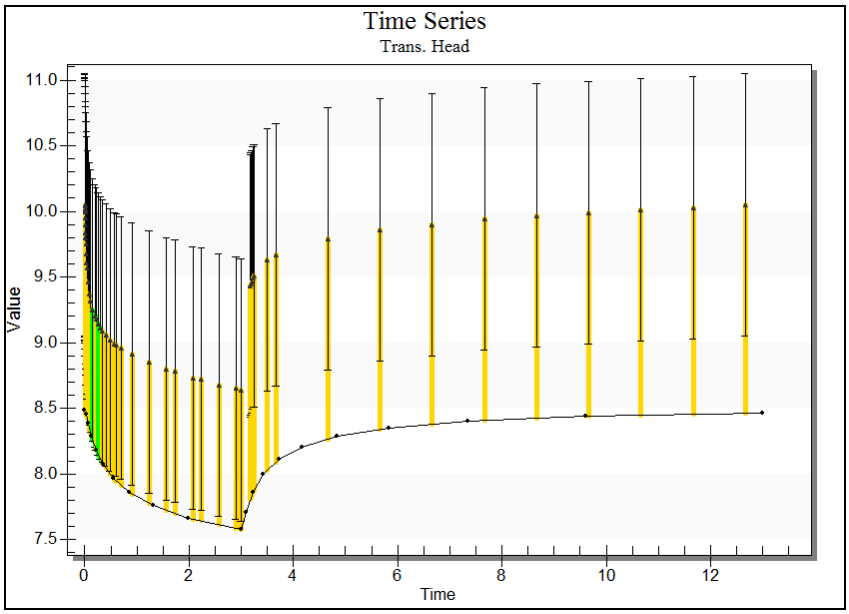


Figure 9 Time series plot after PEST run

It would be desirable to improve the fit between the simulated and the observed heads. The user could adjust some of the PEST input parameters to allow PEST to run more with this model, or the user could try using pilot points with the parameters. In this case, the user will use pilot points to try to get a better fit at all of the wells.

12 Pilot Points

Now the user will import pilot point data. For more information on pilot points, refer to the "MODFLOW – Pest Pilot Points" tutorial.

1. Select the **Open** button.
2. Navigate to the *Tutorials\MODFLOW\trans_pest_pumptest* directory
3. In the *Open* dialog, change the *Files of type* selection to "Text Files (*.txt)."
4. Select the file named "pp.txt."
5. Click **Open**.

The *Text Import Wizard* will open, showing the contents of the file.

6. Turn on the *Heading row* check box.
7. Click the **Next** button.

Pilot points are 2D scatter points, which is the default option. GMS has everything already set up in the desired way.

8. Click the **Finish** button on the second page of the wizard.

12.1 Using Pilot Points with the Parameters

It is necessary to change the MODFLOW parameters so that they will use the pilot points that were just imported.

1. Select the *MODFLOW* | **Parameters** command.
2. For parameter "HK_15," select the drop-down arrow in the *Value* column.
3. Then select "<Pilot points>" from the drop-down list.

The interpolation options associated with the pilot points can be changed by clicking on the small button above the drop-down arrow in the *Value* column.

4. Click on the button above the drop-down arrow in the *Value* column for parameter "HK_15."

This brings up the *2D Interpolation Options* dialog. Here, the user can select the scatter point set and dataset used with the parameter as well as the interpolation scheme.

5. Make sure the *Dataset* is set to "HK_15."
6. Select **OK** to exit the *2D Interpolation Options* dialog.
7. Turn on the "<Pilot points>" option for parameters "HK_30" and **"**RCH_300" by following the steps 2–3 listed above.
8. Click on the button above the drop-down arrow in the *Value* column for parameter for "HK_30,"
9. Make sure the *Dataset* is set to "HK_30."
10. Select **OK** to exit the *2D Interpolation Options* dialog.
11. Click on the button above the drop-down arrow in the *Value* column for parameter for "RCH_300."
12. Make sure the *Dataset* is set to "RCH_300."
13. Select **OK** to exit the *2D Interpolation Options* dialog.
14. When finished, select **OK** to exit the *Parameters* dialog.

13 Running PEST

Now the user will run Parallel PEST again. However, this time the user will use the SVD-Assist option. Currently, the user has 24 pilot points for each parameter. This

means that for each PEST iteration, PEST will run MODFLOW once for each existing pilot point. For this model, that would be a total of 72 MODFLOW runs in addition to runs for the other 3 parameters. Using SVD-Assist, PEST can decrease the number of necessary MODFLOW runs for each PEST iteration.[3]

1. Select the *MODFLOW* | **Parameter Estimation** command to open the *PEST* dialog.
2. Select the check box next to *Use SVD-Assist*.
3. Select **OK** to exit the dialog.
4. Select the *File* | **Save As** menu command.
5. Change the project name to "pumptest_pest_pp.gpr."
6. Click **Save.**
7. Select the **Run MODFLOW** button.

Depending on the speed of the user's computer, Parallel PEST will take about 5 to 20 minutes to run this problem. When finished, you should see that PEST completed 5 iterations and had a final model error of about 19.6.

8. Select the **Close** button.

The contours and time series plot should update. The time series curve for the observation well should look like the next figure. The user has significantly improved the match between the field values and the model simulated values.

3 You will *not* want to complete this portion of the tutorial if you are not running Parallel PEST.

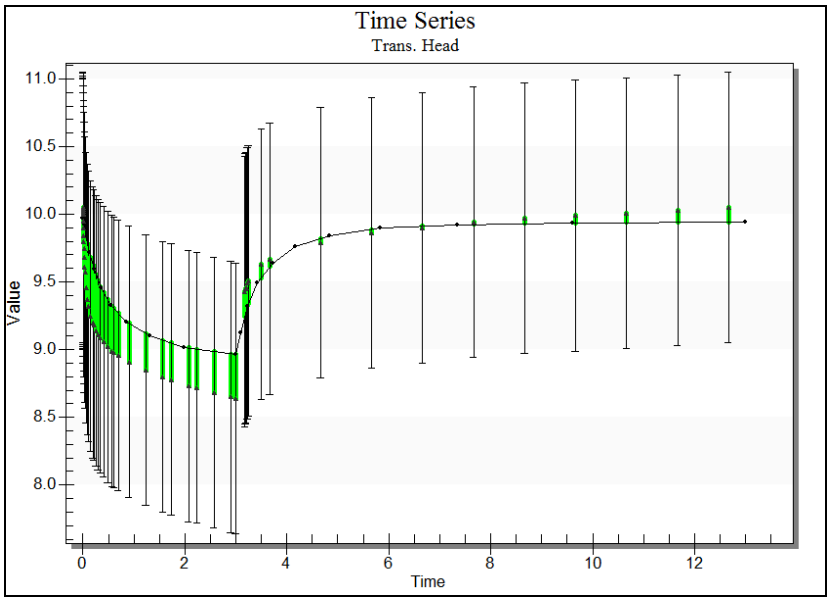


Figure 10 Time series Plot after PEST Calibration

If the user wants to see the final hydraulic conductivity and recharge arrays, look under the "pumptest_pest_pp (MODFLOW)" solution item in the Project Explorer. Find a folder called *Parameters*, and inside that folder, find a dataset named "HK Parameter - 30, -15." Selecting this dataset, the user can see the final hydraulic conductivity array computed from the pilot points. The recharge is also available.

14 Conclusion

This concludes this tutorial. Here are the key concepts in this tutorial:

- The user can import transient observation data with the *Text Import Wizard*.
- It is possible to use PEST to calibrate transient MODFLOW models.
- It is important to have the first stress period as steady state when calibrating a transient model so that the computed heads for the first transient stress period are consistent with the model input parameters.
- It is possible to view the final array values from parameters that use pilot points by selecting the dataset under the MODFLOW solution.

