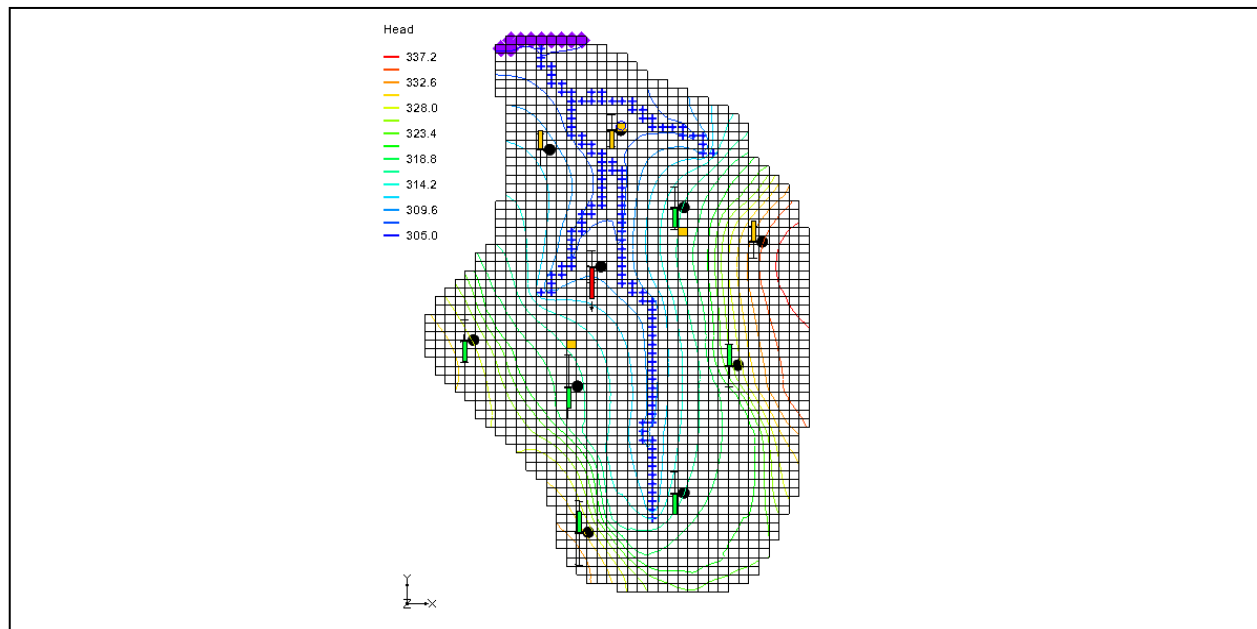*GMS 10.1 Tutorial*

MODFLOW – Model Calibration

Manually vary input values and use observations to calibrate a MODFLOW model

Objectives

GMS contains a suite of tools to assist in the model calibration process, including observation points and different types of plots. These tools are described in this tutorial. A MODFLOW model is manually calibrated by adjusting inputs and watching how the computed values compare to the observed values.

Prerequisite Tutorials	Required Components	Time
• MODFLOW – Conceptual Model Approach I	• Grid Module • Map Module • MODFLOW	• 25-40 minutes

AQUAVEO™

1   Introduction

An important part of any groundwater modeling exercise is the model calibration process. In order for a groundwater model to be used in any type of predictive role, it must be demonstrated that the model can successfully simulate observed aquifer behavior. Calibration is a process wherein certain parameters of the model such as recharge and hydraulic conductivity are altered in a systematic fashion and the model is repeatedly run until the computed solution matches field-observed values within an acceptable level of accuracy.

The model calibration exercise in this tutorial is based on the MODFLOW model. Thus, it is recommended to complete the "MODFLOW – Conceptual Model Approach I" tutorial prior to beginning this tutorial. Although this particular exercise is based on MODFLOW, the calibration tools in GMS are designed in a general purpose fashion and can be used with any model.

1.1   Description of Problem

A groundwater model for a medium-sized basin is shown in Figure 1. The basin encompasses 72.5 square kilometers. It is in a semi-arid climate, with average annual precipitation of 0.381 m/yr. Most of this precipitation is lost through evapotranspiration.

The recharge that reaches the aquifer eventually drains into a small stream at the center of the basin. This stream drains to the north and eventually empties into a lake with elevation 304.8 m. Three wells in the basin also extract water from the aquifer. The perimeter of the basin is bounded by low permeability crystalline rock. There are ten observation wells in the basin. There is also a stream flow gauge at the bottom end of the stream.
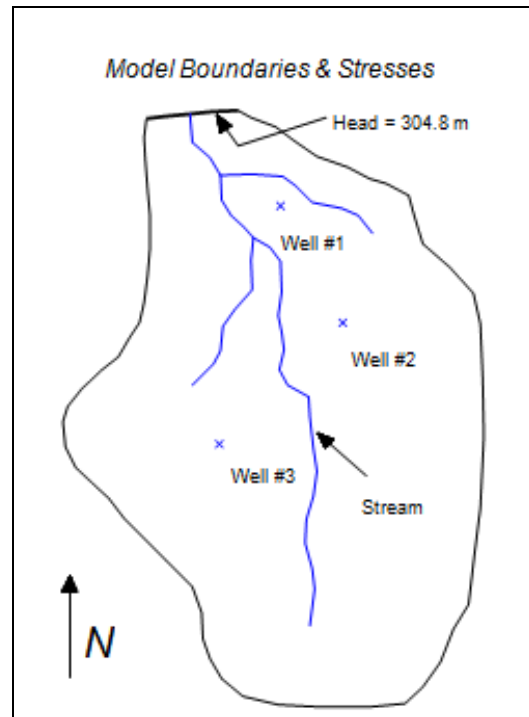

Figure 1   Sample model used in calibration exercise

The assumed recharge and hydraulic conductivity zones for the model are shown in Figure 2. The model region encompasses fractured and weathered bedrock as well as alluvial material, grading from hydraulically tighter materials in the south to more permeable materials in the north. Furthermore, the materials around the stream tend to be coarser, cleaner, and thus more permeable. The topmost region of the model near the lake has a high level of phreatophytic plant life.

The first task of this exercise will be to import a single layer, unconfined MODFLOW model that has been constructed for the site. This model contains an initial estimate of hydraulic conductivities and recharge. A solution computed with this initial model will then be imported and the error in the initial solution will be analyzed. New values for hydraulic conductivity and/or recharge will be entered, a new MODFLOW solution will be generated, and a new error estimate will be computed and plotted. These steps should be repeated until the error is reasonably small.
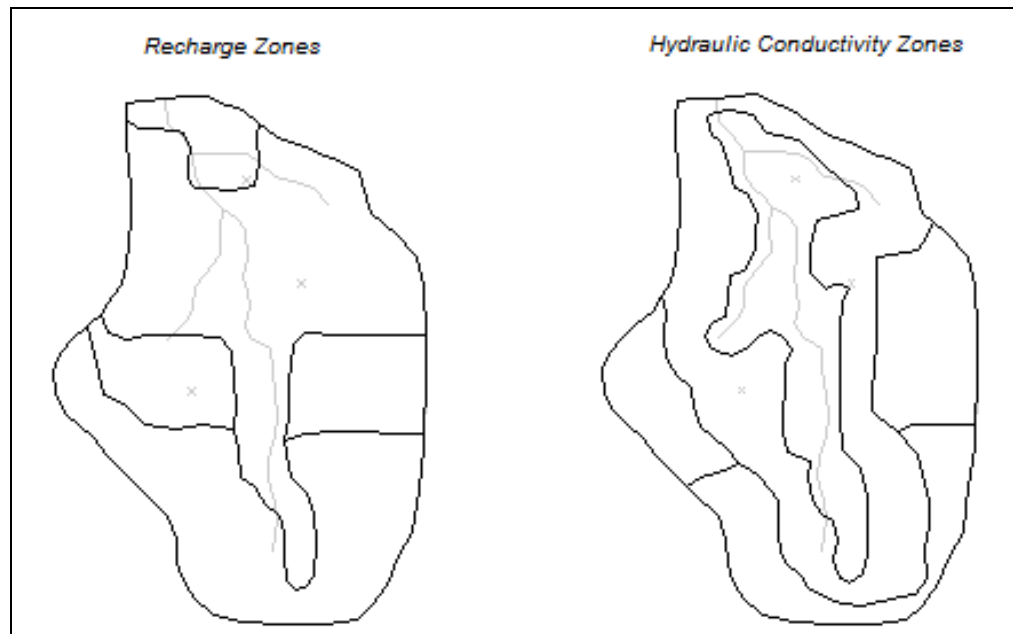
Figure 2   Recharge and hydraulic conductivity zones

1.2   Getting Started

Do the following to get started:

1. If necessary, launch GMS.

2. If GMS is already running, select *File* / **New** to ensure that the program settings are restored to their default state.

2   Importing the Model

First, import the model:

1. Click **Open** 📂 to bring up the *Open* dialog.

2. Select "Project Files (*.gpr)" from the *Files of type* drop-down.

3. Browse to the *Tutorials\MODFLOW\calib* directory and select "bigval.gpr".

4. Click **Open** to import the project and exit the *Open* dialog.

5. Right-click in the Project Explorer and select **Expand All**.

Both a grid and a conceptual model should appear (Figure 3). The conceptual model was used to construct the MODFLOW model and consists of three coverages. The currently active coverage contains the model boundary, the specified head boundary, the stream, and the wells. There is also a coverage of recharge zones and a coverage of hydraulic conductivity zones. For the initial simulation, a single value of hydraulic conductivity

(2.4 m/day) and a single value of recharge (7.62e-5 m/day) have been assigned. The polygonal zones of hydraulic conductivity and recharge will be edited as the tutorial progresses to reduce the model error.
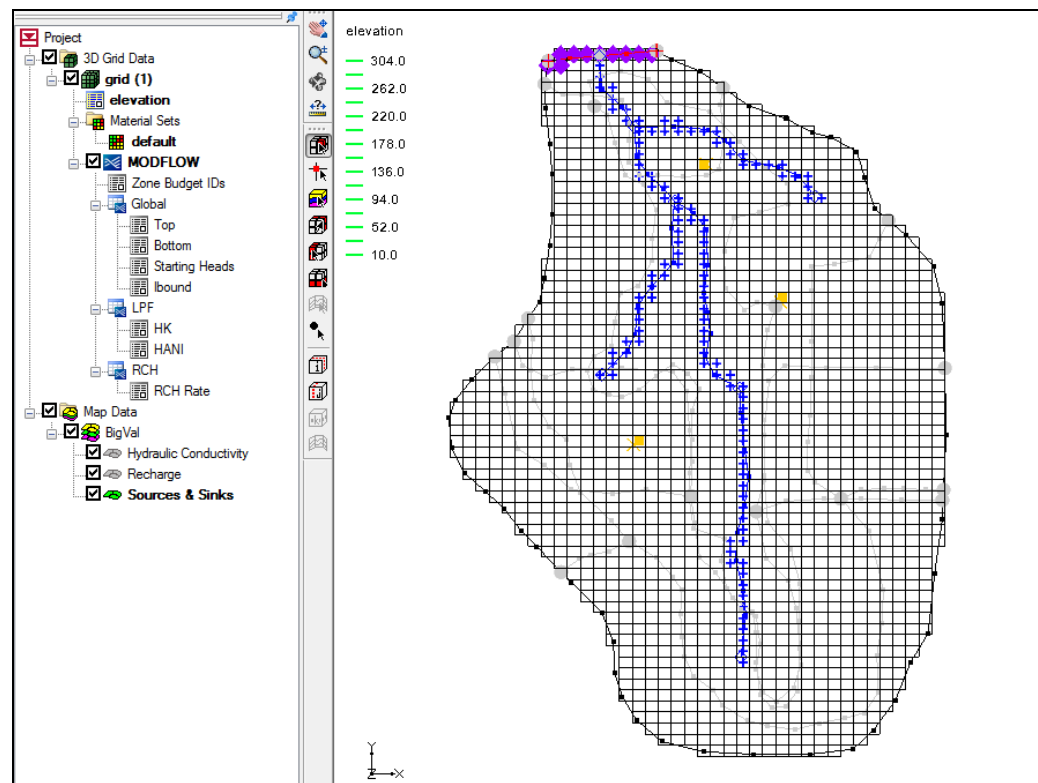

Figure 3   Initial screen showing the model and three coverages

3   Observation Data

The tutorial will be using two types of observation data in the calibration process: water table elevations from observation wells and observed flow rates in the stream. Since the model is in a fairly arid region, assume that most of the flow to the stream is from groundwater flow.

4   Entering Observation Points

First, it is necessary to enter a set of observation points representing the observed head in the ten observation wells in the region. Observation points are created in the Map module.

4.1   Creating a Coverage with Observation Properties

Before entering observation points, first create a coverage with observation properties.

1. Right-click on " BigVal" in the Project Explorer and select **New Coverage…** to bring up the *Coverage Setup* dialog.

2. Enter "Observation Wells" as the *Coverage name*.

3. In the *Observation Points* column, turn on *Head*.

4. Select "By layer number" from the *3D grid layer option for obs. pts* drop-down.

5. Click **OK** to exit the *Coverage Setup* dialog.

4.2   Creating an Observation Point

It is now possible to create an observation point. To create the point, do the following:

1. Select the " Observation wells" coverage to make it active.

2. Using the **Create Point** tool, click once anywhere on the model to create a new point.

3. In the XYZF Bar, enter "4470" for *X* and "9965" for *Y* (Figure 4). This will move the point to those coordinates.

X: 4770.0	Y: 9965.0	Z: 0.0	F:

Figure 4   XYZF Bar

To assign the properties to the point:

4. With the point still selected, click **Properties** to bring up the *Attribute Table* dialog.

5. Enter "Point #1" in the *Name* column of the spreadsheet.

6. Select "obs. pt." from the drop-down in the *Type* column.

7. Enter "305" in the *Obs. Head* column.

8. Enter "1.5" in the *Obs. Head interval* column.

This interval represents the estimated error (±) in the observed value. This confidence value represents the confidence in the error estimate. The interval can be used as a calibration target. Calibration is achieved when the error is within the estimated error interval (± 1.5 m, in this case) of the observed value. In other words, if the computed head falls between 303.5 and 306.5, the calibration target is reached for this observation well.

9. Enter "95" in the *Obs Head conf(%)* column.

10. Click **OK** to close the *Attribute Table* dialog.

4.3   Calibration Target

At this point, a calibration target like the one shown in Figure 5 typically shows up next to the observation point. However, the observation measurement model in this case is MODFLOW. With MODFLOW, the simulation must be run and the solution imported before the calibration targets appear. This is because the computed values are output by MODFLOW and are not calculated internally by GMS.

To make the calibration targets appear, run MODFLOW and import the solution.

1.   Select *File |* **Save As…** to bring up the *Save As* dialog.

2.   Select "Project Files (*.gpr)" from the *Save as type* drop-down.

3.   Enter **"**run2.gpr" as the *File name* and click **Save** to exit the *Save As* dialog.

4.   Select the " 3D Grid Data" folder in the Project Explorer to make it active.

5.   Click **Run MODFLOW** to bring up the *MODFLOW* model wrapper dialog.

6.   When MODFLOW finishes running, turn on *Read solution on exit* and *Turn on contours (if not on already).*

7.   Click **Close** to exit the *MODFLOW* model wrapper dialog.

Note that a calibration target is now drawn next to the point (Figure 5).
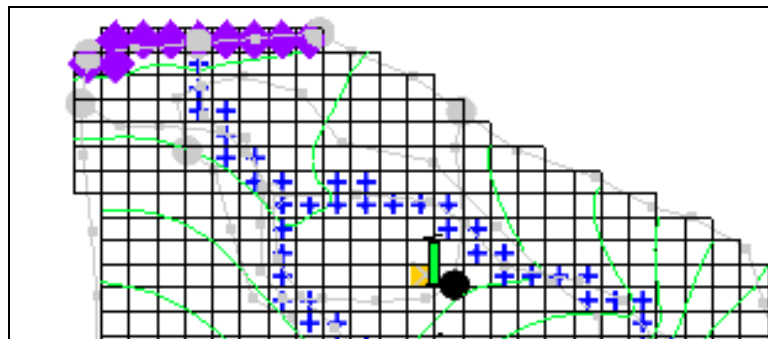

Figure 5   Green error bar on calibration target

The components of a calibration target are illustrated in   Figure 6. The center of the target corresponds to the observed value. The top of the target corresponds to the observed value plus the interval and the bottom corresponds to the observed value minus the interval.

The colored bar represents the error. If the bar lies entirely within the target, the color bar is drawn in green. If the bar is outside the target but the error is less than 200%, the bar is drawn in yellow. If the error is greater than 200%, the bar is drawn in red. In this case, the bar should be green.
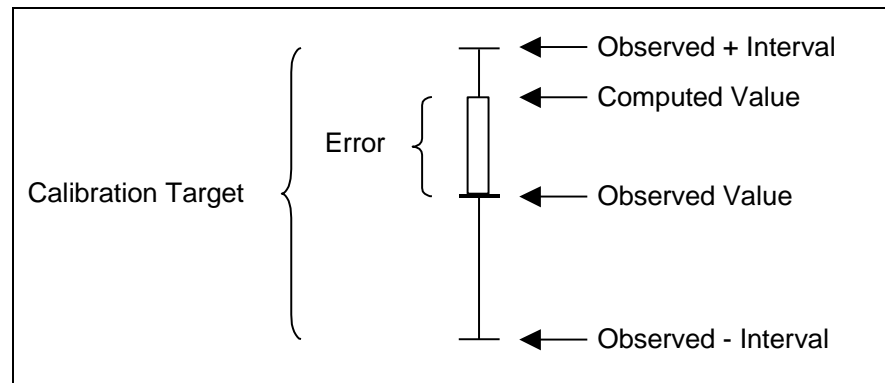
Figure 6   Calibration target

4.4   Point Statistics

More detailed statistics concerning the error at the point can be viewed by selecting the point.

1. Select the " Map Data" folder in the Project Explorer to make it active.

2. Using the **Select Points/Nodes** tool, click on the observation point.

Notice that a set of statistics related to the point is displayed in the Status Bar at the bottom left side of the GMS screen (Figure 7).

ID: 1; Name = Point #1 Computed: 306.337, Observed: 305.000, Residual: -1.3369

Figure 7   Status bar showing information on the selected observation point

5   Reading in a Set of Observation Points

The next step would be to enter the remaining nine observation points. In the interest of time, simply import a previously-prepared map file containing all ten points.

5.1   Deleting the Observation Point

Before importing the map file containing the observation points, delete the point that was created in the *Observation Wells* coverage.

1. Using the **Select Points/Nodes** tool, right-click on the point that was created earlier and select **Delete**.

5.2   Importing the Points

Now import a file containing several observation points.

1. Click **Open** 📂 to bring up the *Open* dialog.

2. Select "Text Files (*.txt;*.csv)" from the *Files of type* drop-down.

3. Select "obswells.txt" and click **Open** to exit the *Open* dialog and bring up the *Step 1 of 2* page of the *Text Import Wizard* dialog.

5.3   Text Import Wizard

Opening a TXT file in GMS will bring up this dialog. In the first step of the wizard, it is necessary to identify the delimiters that separate the data into columns. In the second step of the wizard, map the columns to fields that GMS recognizes.

1. In the section below the *File import options* section, turn on *Heading row*.

2. Click **Next>** to go to the *Step 2 of 2* page of the *Text Import Options* dialog.

3. Select "Observation data" from the *GMS data type* drop-down.

Notice in the file preview section of the dialog that GMS has mapped the different columns of data in the file into fields recognized by GMS. No changes need to be made here.

4. Click **Finish** to import the observation data and close the *Text Import Wizard* dialog.

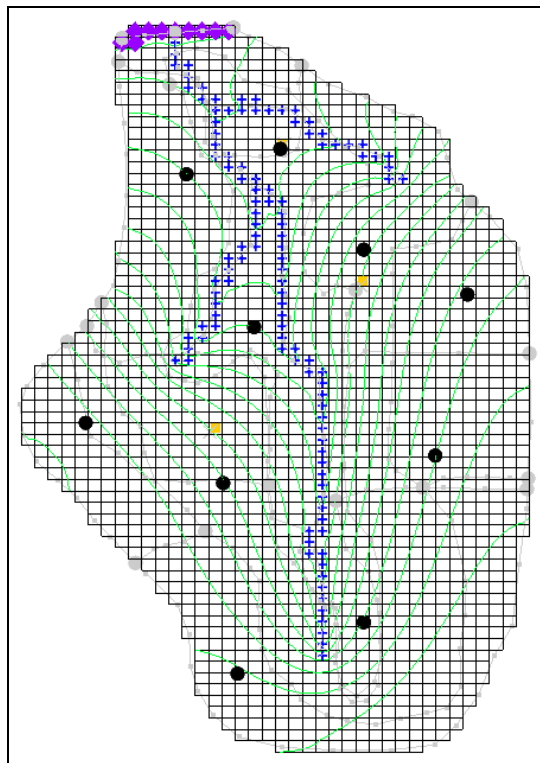Ten observation wells will appear (Figure 8).

Figure 8   The imported observation wells

6   Entering the Observed Stream Flow

Now that the observation points are defined, enter the observed flow in the stream. Observed flows are assigned directly to arcs and polygons in the local source/sink coverage of the conceptual model. Assign an observed flow to the river arcs. MODFLOW determines the computed flow from the aquifer to the stream. This flow value will be compared to the observed flow.

GMS provides two methods for assigning observed flow: to individual arcs or to a group of arcs. The stream flow that was measured at the site represents the total flow from the aquifer to the stream at the stream outlet at the top of the model. This flow represents the flow from the aquifer to the stream for the entire stream network. Thus, assign the observed flow to a group of arcs. When reading a solution, GMS will then automatically sum the computed flow for all arcs in the group.

Before assigning the observed flow, first create an arc group:

1. Right-click on the "🐟 Sources & Sinks" coverage and select **Coverage Setup** to open the *Coverage Setup* dialog.

2. In the *Sources/Sinks/BCs* column, turn on *Observed Flow*.

3. Click **OK** to exit the *Coverage Setup* dialog.

4. Select "Sources & Sinks" 🐟 to make it active.

5. Using the **Select Arcs** 🔏 tool while holding down the *Shift* key, click on each of the river arcs. Be sure to select all six arcs.

6. Select *Feature Object* | **Create Arc Group**.

This creates a new object out of the selected objects. It is now possible to assign an observed flow to the arc group.

7. Using the **Select Arc Groups** 🔏 tool, double-click on any of the river arcs to open the *Attribute Table* dialog.

8. Check the box in the *Obs.flow* column of the spreadsheet.

9. Enter "-4644" in the *Obs. flow rate (m^3/d)* column.

10. Enter "210" in the *Obs. Flow interval* column.

The entered values indicate that, to achieve calibration, the computed flow should be between -4434 and -4854 m^3/day (4644 +/- 210).

11. Click **OK** to exit the *Attribute Table* dialog.

12. Click outside the arc group to unselect it.

13. **Save** 💾 the project.

14. Select *MODFLOW* | **Run MODFLOW** to bring up the *MODFLOW* model wrapper dialog.

15. When the model finishes running, turn on *Read solution on exit* and *Turn on contours (if not on already)*.

16. Click **Close** to exit the *MODFLOW* model wrapper dialog.

At this point, a calibration target should appear for the observed flow on the arc group (Figure 9).
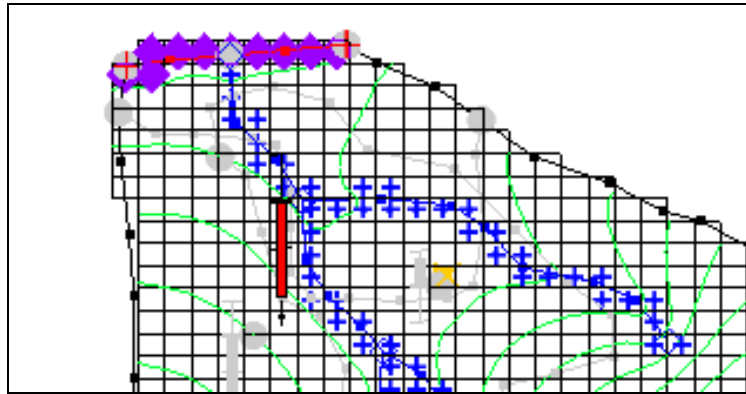

Figure 9   Calibration target for the river arc group observed flow

7   Viewing the Solution

The contours currently shown on the 3D grid are the heads from the MODFLOW run with the optimum parameter values.

7.1   Error Summary

Now look at the observation targets in the map model and the error associated with this model run.

1. Select the "⬤ Sources & Sinks" coverage to make it active.

2. Using the **Select Arc Group** 🏃🗲 tool, select the arc group by clicking on the river arc.

Notice that the computed and observed flow is reported in the status bar at the bottom of the Graphics Window.

3. Right-click on the "📠 run2 (MODFLOW)" solution in the Project Explorer and select **Properties…** to open the *Properties* dialog.

This dialog shows a spreadsheet giving the error from this model run. The spreadsheet shows the error from the head observations, the flow observations, and the combined weighted observations.

4. When finished viewing the properties, click **Done** to close the *Properties* dialog.

7.2   Computed vs. Observed Data Plot

Next, generate some plots related to the calibration error. Create two plots related to the error at the observation points.

1. Click **Plot Wizard** 📖 to open the *Step 1 of 2* page of the *Plot Wizard* dialog.

2. Select "Computed vs. Observed Data" from the list on the left.

3. Click **Next** to go to the *Step 2 of 2* page of the *Plot Wizard* dialog.

4. Click **Finish** to close the *Plot Wizard* dialog.

In the first plot, a symbol is drawn for each of the observation points (Figure 10). A point that plots on or near the diagonal line indicates a low error. Points far from the diagonal have a larger error.
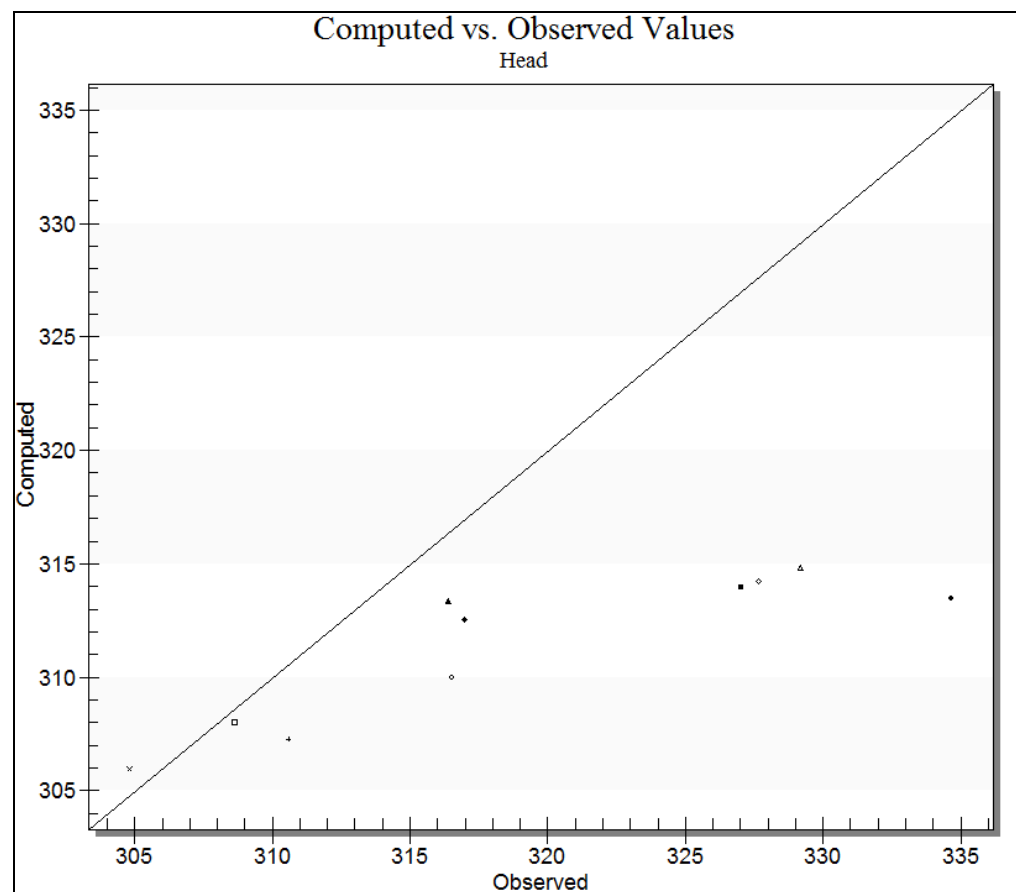

Figure 10   Ten observation points plotted

8   Editing the Hydraulic Conductivity

The next step in the calibration exercise is to change the model parameters and re-run the model. Note that the errors are mostly red and negative on the "Observation Wells"

coverage. This indicates that the observed value is much larger than the current computed value. Begin by changing the hydraulic conductivity in these zones. Edit the hydraulic conductivity by changing the hydraulic conductivity assigned to the polygonal zones in the conceptual model.

Before editing the hydraulic conductivity values, make the hydraulic conductivity zone coverage the active coverage.

1. Select the "🐢 Hydraulic Conductivity" coverage to make it active.

To edit the hydraulic conductivity values:

2. Using the **Select Polygons** 🔲 tool while holding the *Shift* key, select polygons 1 and 2 as shown in   Figure 11.

3. Click **Properties** 🖳 to open the *Attribute Table* dialog.

4. On the *All* row, enter "0.6" in the *Horizontal K (m/d)* column. This changes the value for both polygons at the same time.

5. Click **OK** to close the *Attribute Table* dialog.

6. Double-click on polygon 3 as shown in   Figure 11 to open the *Attribute Table* dialog.

7. Enter "0.15" in the *Horizontal K* column.

8. Click **OK** to close the *Attribute Table* dialog.

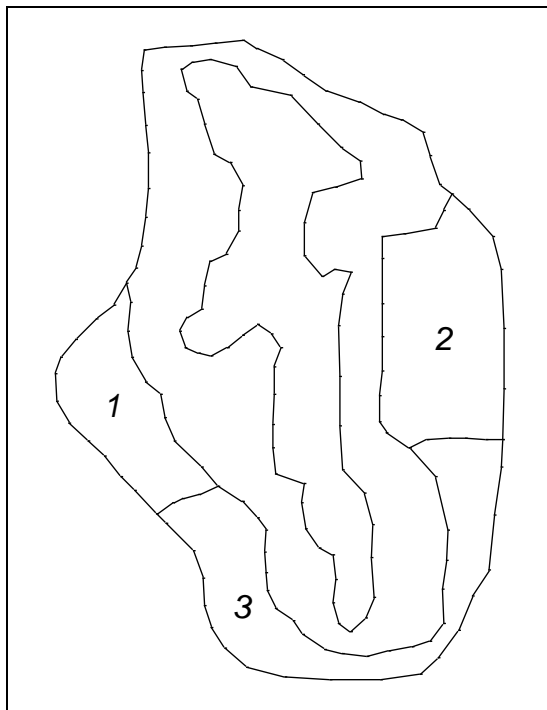9. Click outside the model to unselect the polygon.

Figure 11   Polygons to be selected.

---

9   Converting the Model

Now that the values have been edited, the next step is to convert the conceptual model to the grid-based numerical model:

1.   Click **Map → MODFLOW** 🔲 to bring up the *Map → Model* dialog.

2.   Select *All applicable coverages* and click **OK** to close the *Map → Model* dialog.

10   Computing a Solution

The next step is to save the MODFLOW model with the new values and compute a new solution.

10.1   Saving the Simulation

To save the simulation:

1.   Select *File | **Save As…*** to bring up the *Save As* dialog.

2.   Select "Project Files (*.gpr)" from the *Save as type* drop-down.

3.   Enter "run3.gpr" as the *File name*.

4.   Click **Save** to close the *Save As* dialog.

10.2   Running MODFLOW

Do the following to run MODFLOW:

1.   Click **Run MODFLOW** 🔲 to bring up the *MODFLOW* model wrapper dialog.

2.   When the MODFLOW simulation is completed, turn on *Read solution on exit* and *Turn on contours (if not on already)*.

3.   Click **Close** to import the solution and close the *MODFLOW* model wrapper dialog.

Note that the plot in the *Plot Window* has been updated (Figure 12). Although the error improved for the observation wells, the head is still too low on the left and right sides of the model. Up to this point, the tutorial has not paid much attention to the flow target on the arc group. In the next section, create a plot that shows how well the flow target is being met.
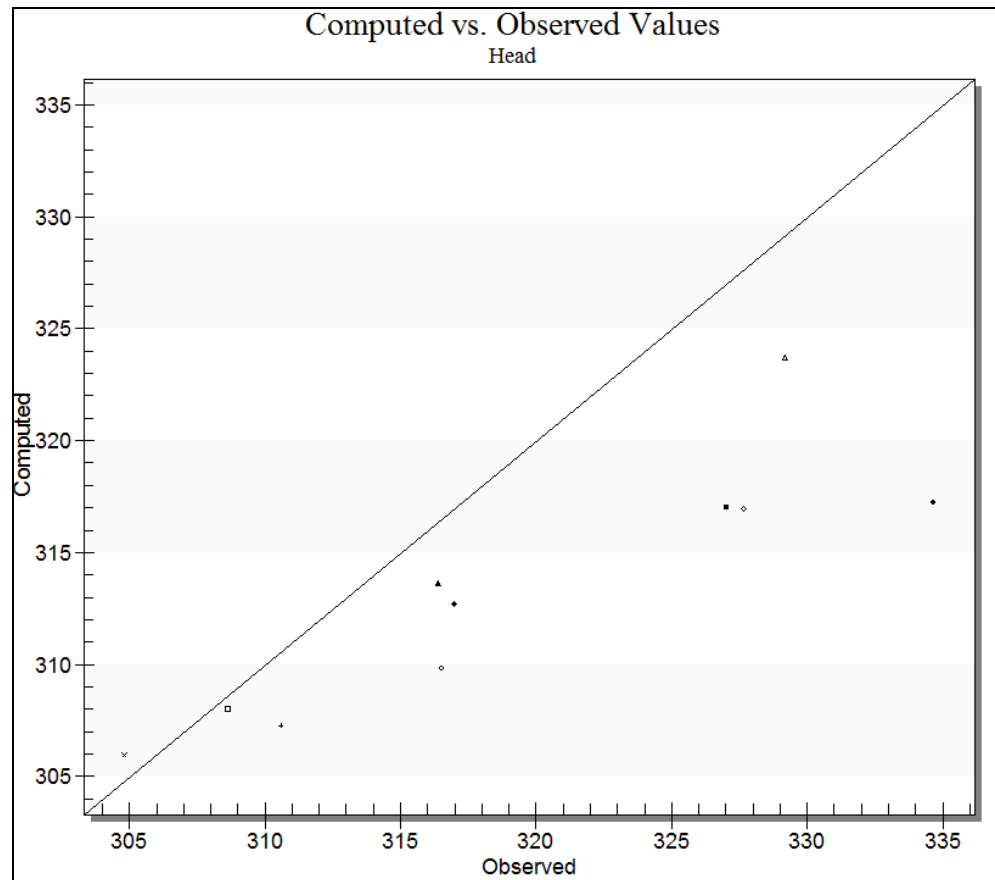
Figure 12   Notice how several of the points are closer to the line

11   Error vs. Simulation Plot

When performing trial and error calibration, it is important to keep track of the trend in the error as new solutions are repeatedly computed. GMS provides a special calibration plot to simplify this task.

1.   Click **Plot Wizard** 📊 to bring up the *Step 1 of 2* page of the *Plot Wizard* dialog.

2.   Select "Error vs. Simulation" from the list on the left.

3.   Click **Next** to go to the *Step 2 of 2* page of the *Plot Wizard* dialog.

4.   Click **Finish** to close the *Plot Wizard* dialog.

The change in each of the three error norms from one solution to the next is clearly visible in the new plot (Figure 13). The plot is updated as each new solution is imported. Ideally, the error should gradually become smaller and smaller as changes are made to the model. If a bad choice is made in changing the model, the error may temporarily increase. In most cases, it is a good idea to keep a log of the changes made and the resulting errors.

5.   When finished viewing the plot, close the plot window.
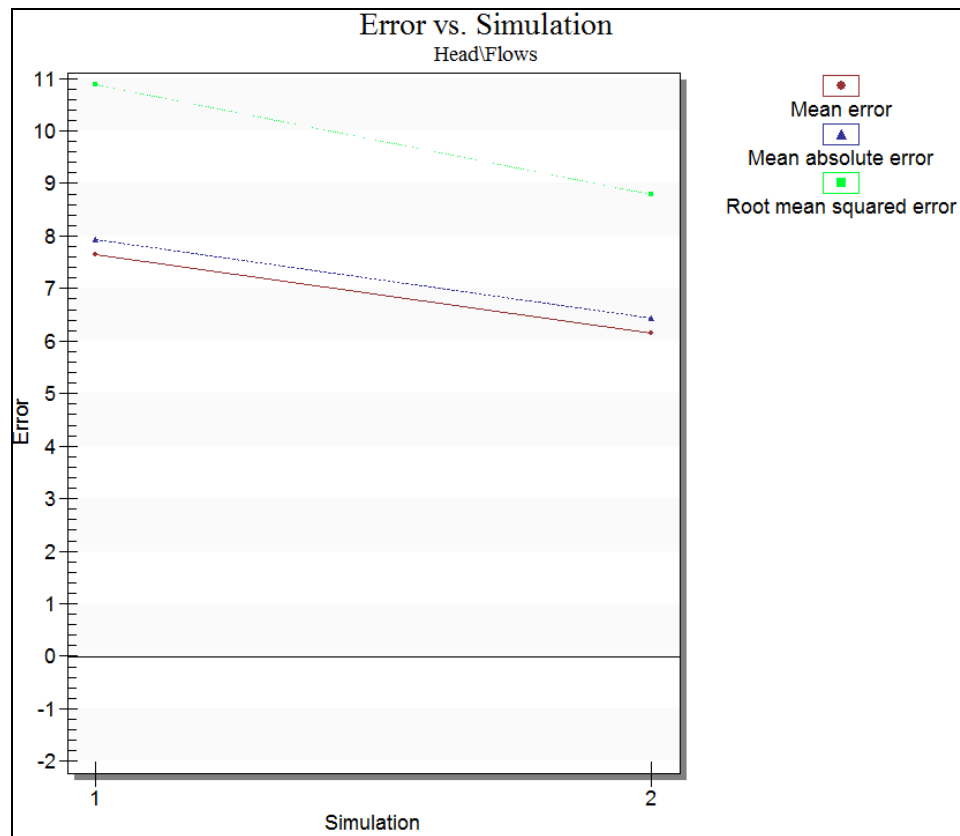
6. Maximize the graphic window again.

Figure 13   The change in error is more clearly visible

12   Continuing the Trial and Error Calibration

At this point, continue with the trial-and-error calibration process using the steps outlined above. If changing the recharge and the hydraulic conductivity values, be sure to make the coverage visible by selecting it in the Project Explorer.

12.1   Changing Values vs. Changing Zones

For this tutorial, a good match between the computed and observed values can be obtained simply by changing the hydraulic conductivity and recharge values assigned to the polygonal zones. In a real application, however, the size and distribution of the zones may need to be changed in addition to the values assigned to the zones.

12.2   Viewing the Answer

To view the "answer," a map file can be imported that contains a set of parameter values that result in a solution that satisfies the calibration target for each of the ten observation wells.

Before importing the new conceptual model, delete the "BigVal" conceptual model:

1.   Right-click on the "BigVal" 🗿 conceptual model and select **Delete**.

To import the new conceptual model:

2.   Click **Open** 📂 to bring up the *Open* dialog.

3.   Select "Project Files (*.gpr)" from the *Files of type* drop-down.

4.   Select "answer.gpr".

5.   Turn on *Import into current project*.

6.   Click **Open** to import the project and close the *Open* dialog.

This model can now be converted to the grid, and a new solution can be computed using the steps described above.

13   Conclusion

This concludes the "MODFLOW – Model Calibration" tutorial. The following key topics were discussed and demonstrated:

* Observation points will display a target next to them that shows how well the computed values match the observed values.

* With MODFLOW, the computed values at the observation points come from MODFLOW and not GMS. Thus, if creating an observation point, MODFLOW will have to be run before seeing a target.

* If wanting to specify a flow observation that applies to a network of streams, create an arc group.

* A number of different plots are available when doing model calibration.

* It is possible to import data from another GMS project by turning on *Import into current project* in the *Open* dialog after selecting the desired project.

* Whenever changes are made to the conceptual model, the *Feature Objects* / **Map → MODFLOW** command must be used and the project must be saved before running MODFLOW.