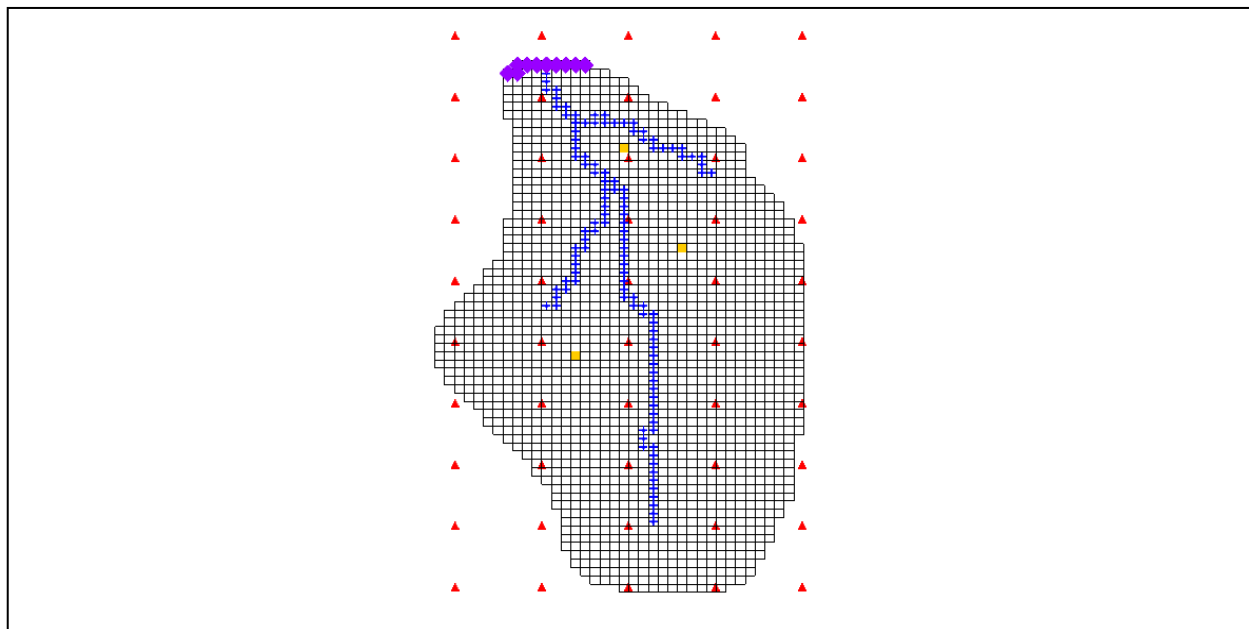


GMS 10.4 Tutorial

MODFLOW – Advanced PEST Pilot Points

Edit regularization options and use more than one set of pilot points to calibrate a MODFLOW model



Objectives

This tutorial demonstrates the features and options related to pilot points when used with PEST. It will review using fixed value pilot points, regularization, and multiple parameters.

Prerequisite Tutorials

- MODFLOW – PEST Pilot Points

Required Components

- Grid Module
- Geostatistics
- Map Module
- MODFLOW
- Inverse Modeling

Time

- 20–30 minutes



1	Introduction	2
1.1	Getting Started	3
2	Importing the Project	3
3	Changing the Regularization Option	4
3.1	Saving the Project	4
3.2	Prior Information Equations	5
3.3	Running PEST	5
4	Multiple Parameters using Pilot Points	6
4.1	Creating a Second Set of Pilot Points.....	7
4.2	Creating the New HK Parameter	8
4.3	Using Pilot Points with RCH Parameter.....	9
4.4	Saving the Project and Running PEST	10
5	A Note on Highly Parameterized Models	11
6	Conclusion	11

1 Introduction

Pilot points can be thought of as a 2D scatter point set. Instead of creating a zone and having the inverse model estimate one value for the entire zone, the value of the parameter within the zone is interpolated from the pilot points. The inverse model then estimates the values at the pilot points. Using pilot points will vary values from cell to cell. When the inverse model runs, the values at the pilot points are adjusted and reinterpolated to the grid cells until the objective function is minimized.

PEST provides an option for the pilot point method called regularization. Regularization imposes an additional measure of constraint to the parameter being interpolated. This constraint is imposed by providing PEST with additional information about the parameter in the form of prior information equations. This constraint makes the inversion process much more stable and makes it possible to violate one of the typical constraints associated with parameter estimation: namely, the requirement that the number of parameters must be less than the number of observations.

With regularization, the number of parameters can greatly exceed the number of observations. As a result, complex hydraulic conductivity distributions can be defined, resulting in extremely low residual error. The pilot point method with regularization is an incredibly powerful feature of PEST.

There are two methods available in GMS for defining the prior information equations for PEST. The two methods can be used simultaneously but are usually used separately. The first method is *Preferred homogeneous regularization*. When this option is selected, the prior information equations written to the PEST control file relate the pilot points to one another. These equations indicate to PEST that—in the absence of any strong influence from the PEST objective function—pilot points that are near to one another should have about the same value.

The second method of regularization is *Preferred value regularization*. When this option is selected, prior information equations written to the PEST control file relate the pilot points to their starting value. These equations indicate to PEST that—in the absence of any strong influence from the PEST objective function—the pilot point values should be

equal to their starting value. Depending on the particular problem being solved, one method may be preferable over the other.

The model to be calibrated in this tutorial is the same model featured in the “MODFLOW – PEST Pilot Points” tutorial. The model includes observed flow data for the stream and observed heads at a set of scattered observation wells. The conceptual model for the site consists of a set of recharge and hydraulic conductivity zones. These zones will be marked as parameters and an inverse model will be used to find a set of recharge and hydraulic conductivity values that minimize the calibration error.

This tutorial will discuss and demonstrate the following:

- Opening a MODFLOW model and solution.
- Changing regularization options and running PEST.
- Using pilot points on different zones and different parameter types.


1.1 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 Project

First, import the modeling project:

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 *pilotpointsAdvanced* directory and select “mfpest_pilot_fixed.gpr”.
4. Click **Open** to import the project and exit the *Open* dialog.

The initial MODFLOW model should appear similar to Figure 1.

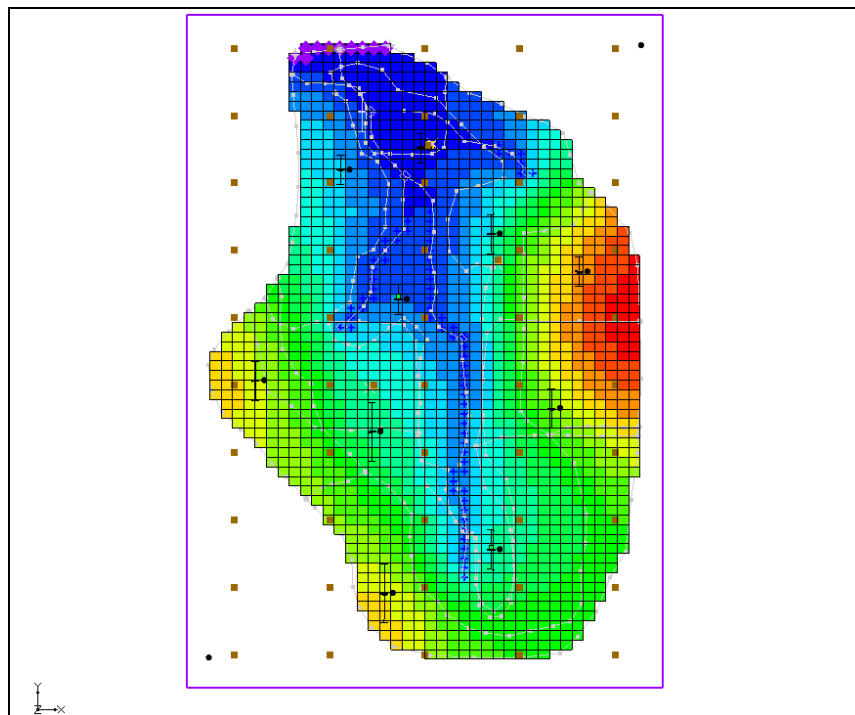


Figure 1 The initial MODFLOW model

3 Changing the Regularization Option

First, the regularization option needs to be changed to use the *Preferred value regularization* method.

1. Select *MODFLOW / Parameter Estimation...* to open the *PEST* dialog.
2. In the *Tikhonov regularization (only use with pilot points)* section, turn off *Preferred homogeneous regularization*.
3. Turn on *Preferred value regularization*.
4. Click **OK** to close the *PEST* dialog.

3.1 Saving the Project

Before running MODFLOW, the project should be saved.

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 “mfpest_pilot_pref_val.gpr” as the *File name*.
4. Click **Save** to close the *Save As* dialog.

3.2 Prior Information Equations

If desired, compare the different prior information equations written from GMS by looking at the RPF files in the MODFLOW directories using a text editor. The prior information equations for *Preferred homogenous regularization*, found in “mfpest_pilot_fixed.rpf”, should look similar to Figure 2.

```
pi0 1.0 * log(sc1v1) - 1.0 * log(sc1v2) = 0.0 0.001251088578 regul_1
pi1 1.0 * log(sc1v1) - 1.0 * log(sc1v3) = 0.0 0.0014852822049 regul_1
pi2 1.0 * log(sc1v1) - 1.0 * log(sc1v4) = 0.0 7.48949022e-006 regul_1
pi3 1.0 * log(sc1v1) - 1.0 * log(sc1v5) = 0.0 5.69586969e-006 regul_1
```

Figure 2 Prior information equations from preferred homogenous regularization


Notice that these equations define a relationship between the different pilot points. In addition, the weight applied to these equations changes as shown by the last number written to each line. In contrast, the prior information equations for *Preferred value regularization*, found in “mfpest_pilot_pref_val.rpf” as shown in Figure 3, have the same weight applied and define a preferred value for each point.

```
pi0 1.0 * log(sc1v1) = -1.313978552818 1.0 regul_1
pi1 1.0 * log(sc1v2) = -1.870689034462 1.0 regul_1
pi2 1.0 * log(sc1v3) = -1.130823969841 1.0 regul_1
pi3 1.0 * log(sc1v4) = -1.1475918293 1.0 regul_1
```

Figure 3 Prior information equations from preferred value regularization

3.3 Running PEST

With the project saved, the next step is to run PEST.

1. Click **Run MODFLOW**  to bring up the *MODFLOW/PEST Parameter Estimation* dialog.

PEST may take several minutes to run, depending on the speed of the computer being used. Once PEST is finished, import the solution.

2. Turn on *Read solution on exit* and *Turn on contours (if not on already)*.
3. Click **Close** to exit the *MODFLOW/PEST Parameter Estimation* dialog.
4. Select *MODFLOW / Parameters...* to open the *Parameters* dialog.
5. Click **Import Optimal Values** to bring up the *Open* dialog.
6. Select “PAR File (*.par,*.bpa)” from the drop-down to the right of the *File name* field.
7. Browse to the *pilotpointsAdvanced\mfpest_pilot_pref_val_MODFLOW* directory and select “mfpest_pilot_pref_val.par”.
8. Click **Open** to exit the *Open* dialog.
9. Click **OK** to close the *Parameters* dialog.

The project should appear similar to Figure 4.

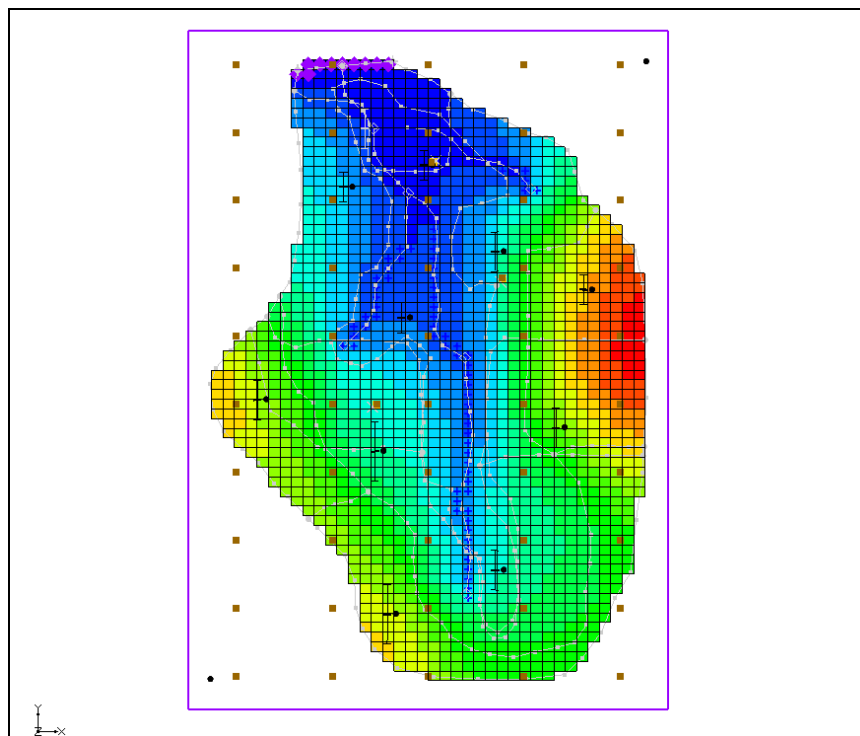






Figure 4 Optimal values

4 Multiple Parameters using Pilot Points

In GMS, pilot points can be used with HK and RCH parameters. Multiple HK (or RCH) parameters that use the same or different pilot points can exist. Create a second HK parameter and a new set of pilot points.

1. In the Project Explorer, turn off the “ 3D Grid Data” folder.
2. Select the “ Hydraulic Conductivity” coverage to make it active.
3. Using the **Select Polygons**  tool, double-click the polygon surrounding the river in the middle of the model to bring up the *Attribute Table* dialog.
4. Enter “-60.0” the *Horizontal K (m/d)* column and click **OK** to close the *Attribute Table* dialog.
5. Right-click on “ Hydraulic Conductivity” and select **Map To / MODFLOW/MODPATH**.

4.1 Creating a Second Set of Pilot Points

Now to create another scatter point set for the new HK parameter by creating a 2D grid and converting it to a scatter point set.

1. Right-click on an empty space in the Project Explorer and select *New / 2D Grid...* to open the *Create Finite Difference Grid* dialog.
2. In the *X-Dimension* section, enter “2240.0” for the *Origin*, “4075.0” for the *Length*, and “3” for the *Number of cells*.
3. In the *Y-Dimension* section, enter “700.0” for the *Origin*, “11100.0” for the *Length*, and “5” for the *Number of cells*.
4. In the *Z-Dimension* section, enter “0.5” as the *Origin*. This value will be assigned to the scatter points created from the grid.
5. In the *Orientation/type* section, select “Cell centered” from the *Type* drop-down.

The dialog should appear similar to Figure 5.

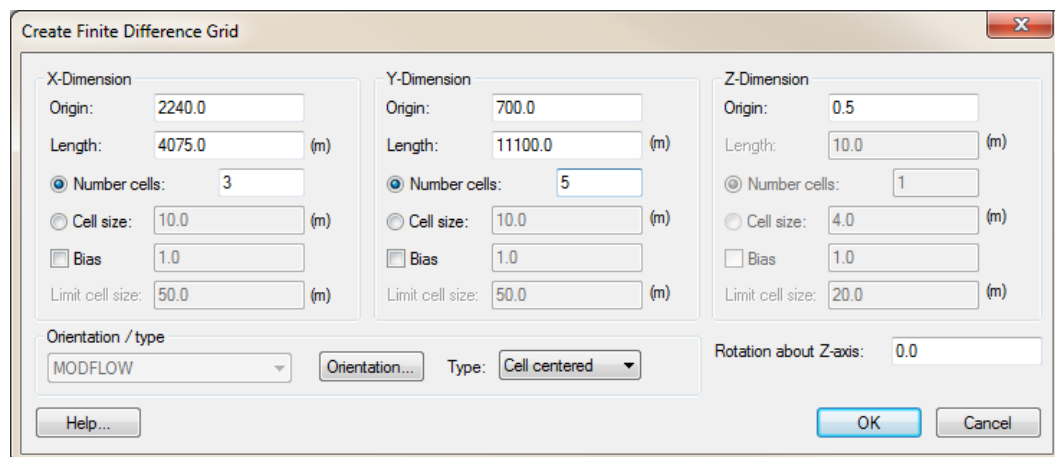




Figure 5 Create Finite Difference Grid dialog

6. Click **OK** to close the *Create Finite Difference Grid* dialog.
7. Right-click on “ grid” and select *Convert to / 2D Scatter Points* to bring up the *Scatter Point Set Name* dialog.
8. Enter “HK_60” as the *New scatter point set name* and click **OK** to close the *Scatter Point Set Name* dialog.
9. Right-click on the same “ grid” item and select **Delete**.

The new scatter points should appear interspersed with the previous scatter points (Figure 6). Feel free to turn on and off each set of scatter points to more clearly see their locations.

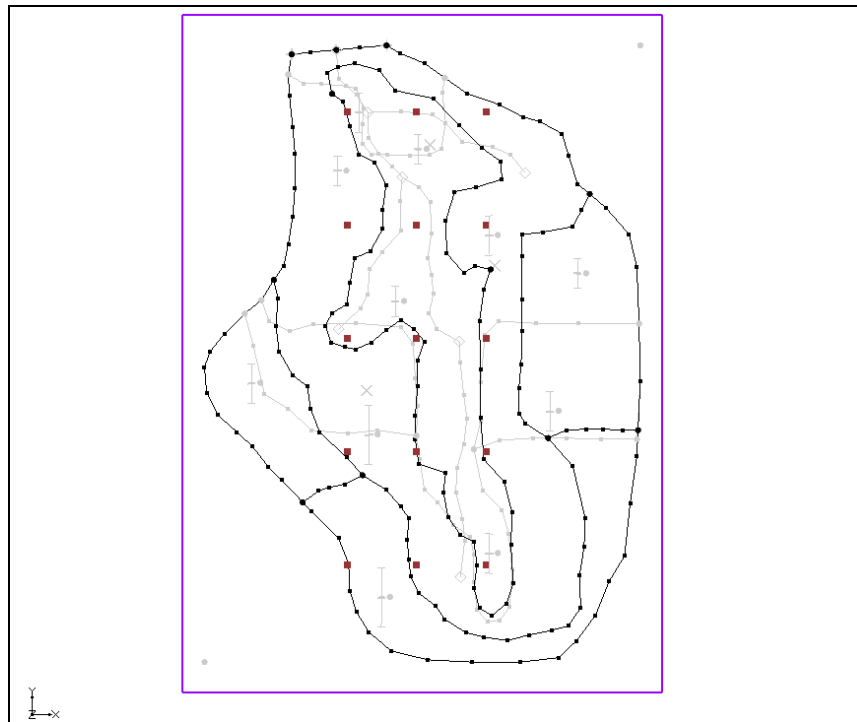


Figure 6 New scatter points (“HK” scatter set turned off)

4.2 Creating the New HK Parameter

A new parameter for the newly created pilot points should now be created.



1. Select *MODFLOW* / **Parameters...** to open the *Parameters* dialog.
2. Click **Initialize From Model**.

Notice that a line for “HK_60” is added to the bottom of the list of parameters.

3. Check the boxes in the *Param. Est. Solve* and *Log Xform* columns on the “HK_60” row.
4. Select “<Pilot points>” from the drop-down in the *Value* column on the “HK_60” row.
5. Click on the button above the drop-down in the *Value* column for parameter “HK_60” to open the *2D interpolation Options* dialog.
6. In the *Interpolating from* section, select “HK_60 (active)” from the *Object* drop-down.
7. Click **OK** to exit the *2D interpolation Options* dialog.
8. Click **OK** to exit the *Parameters* dialog.


4.3 Using Pilot Points with RCH Parameter

Use pilot points to estimate recharge with the model. For the RCH parameter, use the same set of pilot points that the “HK_30” parameter uses, but create a new dataset with starting values for the RCH parameter.

1. Right-click the “ Recharge” coverage and select **Attribute Table...** to bring up the *Attribute Table* dialog.
2. Select “Polygons” from the *Feature type* drop-down.
3. In the *All* row, enter “-150.0” in the *Recharge rate (m/d)* column.
4. Click **OK** to close the *Attribute Table* dialog.
5. Right-click on the “ Recharge” coverage and select *Map To / MODFLOW/MODPATH*.




Creating New Starting Values for the RCH Parameter

Next, create a new dataset on the HK scatter set to provide the starting values for the RCH parameter.

1. Select the “ HK” scatter point set in the Project Explorer to make it active.
2. Select *Edit / Dataset Calculator* to bring up the *Data Calculator* dialog.
3. Enter “1e-5” in the *Expression* field.
4. Enter “RCH” in the *Result* field.
5. Click **Compute** to create a new dataset with all values equal to “1.0e-5”.
6. Click **Done** to exit the *Data Calculator* dialog.

Editing the RCH Parameters

The RCH parameters should now be edited to use pilot points.

1. Select *MODFLOW / Parameters...* to bring up the *Parameters* dialog.
2. Click on “RCH_180” in the *Name* column and click **Delete**.
3. Repeat step 2 for the “RCH_210” parameter row.
4. In the *Value* column for the “RCH_150” row, click the drop-down  and select “<Pilot points>”. This turns on the pilot points option for this parameter.
5. Click on the  button above the drop down  in the *Value* column to bring up the *2D Interpolation Options* dialog.
6. In the *Interpolating from* section, select “HK (active)” from the *Object* drop-down, and “RCH” from the *Dataset* drop-down.


7. Click **OK** to close the *2D Interpolation Options* dialog.
8. Click **OK** to close the *Parameters* dialog.

4.4 Saving the Project and Running PEST


Before running PEST, the project should be saved.

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 “mfpest_pilot_2zones.gpr” as the *File name*.
4. Click **Save** to close the *Save As* dialog.

When pilot points are assigned to both HK and RCH parameters, the prior information equations for the HK and RCH parameters are assigned to different regularization groups. This helps PEST to differentiate weighting among pertinent prior information equations. In other words, PEST works better with this option.

5. Click **Run MODFLOW**  to bring up the *MODFLOW/PEST Parameter Estimation* dialog.

PEST may take several minutes to run, depending on the speed of the computer being used. Once PEST is finished, import the solution.

6. Turn on *Read solution on exit* and *Turn on contours (if not on already)*.
7. Click **Close** to exit the *MODFLOW/PEST Parameter Estimation* dialog.
8. Select *MODFLOW / Parameters...* to open the *Parameters* dialog.
9. Click **Import Optimal Values** to bring up the *Open* dialog.
10. Select “PAR File (*.par,*.bpa)” from the drop-down to the right of the *File name* field.
11. Browse to the *Tutorials\MODFLOW\pilotpointsAdvanced\mfpest_pilot_2zones\MODFLOW* directory and select “mfpest_pilot_2zones.par”.
12. Click **Open** to exit the *Open* dialog.
13. Click **OK** to close the *Parameters* dialog.
14. Turn on “ 3D Grid Data”.

The solution should appear similar to Figure 7. Feel free to compare the solutions by alternating between them in the Project Explorer.

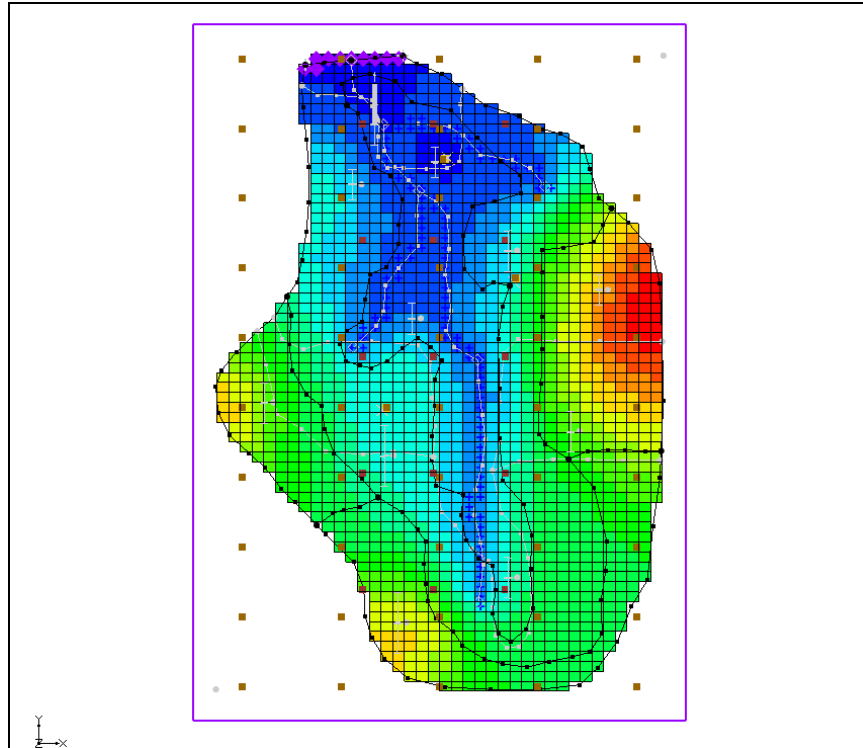


Figure 7 After the final run

5 A Note on Highly Parameterized Models

The model that was just created has over 100 parameters. This is a fairly simple MODFLOW model that converges rather quickly. Most real world problems take longer to run, so it may not be practical to run MODFLOW over 100 times for each PEST iteration.

However, PEST supports a very innovative method known as SVD-Assist which can dramatically reduce the number of model runs required for each PEST iteration. When combining SVD-Assist with Parallel PEST, it becomes practical to use PEST with models containing hundreds or even thousands of parameters. Learn more about these methods in the “MODFLOW – Advanced PEST” tutorial.

6 Conclusion

This concludes the “MODFLOW – Advanced PEST Pilot Points” tutorial. The following key topics were discussed and demonstrated:

- Two different regularization options are available for pilot points: preferred homogeneous and preferred value.
- Multiple sets of pilot points can be used with the same MODFLOW model.