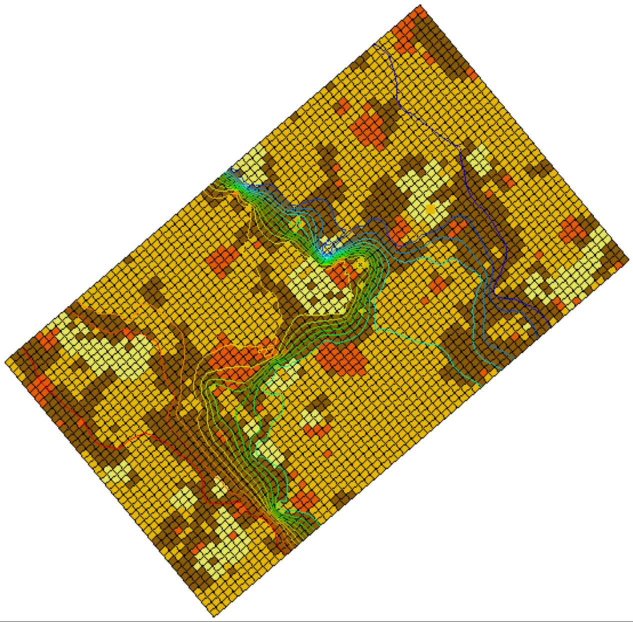
GMS 9.2 Tutorial

MODFLOW – Stochastic Modeling, Indicator Simulations

Use T-PROGS to create multiple material sets and run MODFLOW stochastically



Objectives

The indicator simulation approach with T-PROGS is described. With the indicator simulation approach, multiple equally probable "realizations" of the aquifer stratigraphy are generated. These realizations represent different distributions of material (indicator) zones within the aquifer. A set of aquifer properties is associated with the materials and the model is run once for each of the N realizations.

Prerequisite Tutorials

- None

Required Components

- Grid
- Map
- MODFLOW
- Stochastic Modeling

Time

- 30-60 minutes

1 Contents

2 Introduction

GMS supports three methods for performing stochastic simulations: parameter randomization, indicator simulations, and PEST Null Space Monte Carlo. The indicator simulation approach is described in this tutorial. With the indicator simulation approach, multiple equally probable "realizations" of the aquifer stratigraphy are generated. These realizations represent different distributions of material (indicator) zones within the aquifer. A set of aquifer properties is associated with the materials and the model is run once for each of the N realizations.

In GMS, the multiple realizations of the aquifer heterogeneity are typically generated using the T-PROGS software. T-PROGS can be used to generate two types of output: multiple material sets (arrays of material ids), or multiple MODFLOW HUF input sets. In each case, GMS can launch MODFLOW in batch mode and generate a flow solution for each model instance. For this tutorial we will be using a pre-defined set of material sets generated by T-PROGS. The steps involved in running a T-PROGS simulation are described in the *T-PROGS* tutorial.

2.1 Outline

This is what you will do:

1. Open a MODFLOW model.
2. Edit parameters and run MODFLOW in stochastic mode.
3. Create pathlines.
4. Do a probabilistic capture zone analysis.

3 Description of Problem

The model for this tutorial is based on the Longhorn Army Ammunition Production (LHAAP) site in Texas used in the *TPROGS* tutorial. While we will be using the same site boundaries, the grid used in this simulation will have only one layer to facilitate a shorter run time. We will use a pre-defined T-PROGS simulation containing 30 material sets. The material sets will be used with the stresses and boundary conditions depicted in Figure 1. There is a small drinking water well on the right side of the model. This well averages 25 m^3/d. The regional ground water flow is from left to right. We will use specified head boundaries at the left and right edges of the model at 180 m and 170 m, respectively.

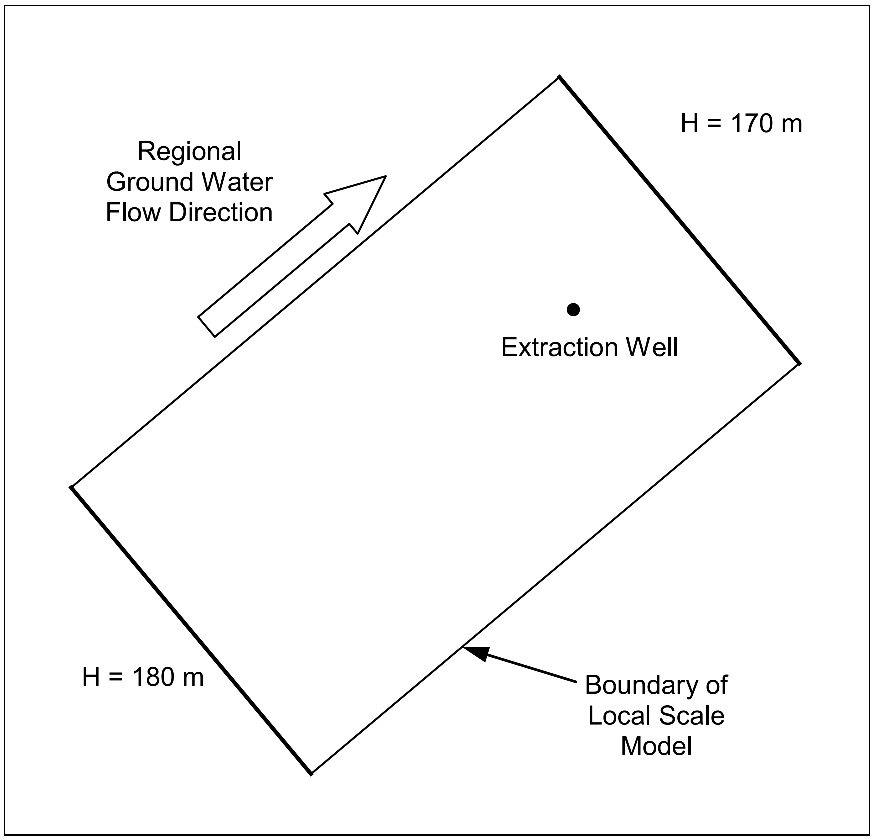


Figure 1. Conceptual representation of site to be modeled.

Once the model is set up, we will run MODFLOW in stochastic mode to generate 30 solutions. We will then run the *Risk Analysis Wizard* to generate a data set representing probability of capture by the extraction well. We will contour this data set to illustrate capture zone boundaries for different probabilities of capture.

4 Getting Started

Let's get started.

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

5 Reading in the Project

First, we will read in a project containing the MODFLOW model and the material sets generated by T-PROGS:

1. Select the *Open* button.
2. Locate and open the **Tutorials\MODFLOW\sto_matset** directory.
3. Open the file entitled **lhaap.gpr**.

You should see a one layer MODFLOW model rotated at a 40 degree angle showing a four-material distribution. We will now view the different material sets generated by T-PROGS:

4. Expand the following items in the *Project Explorer*: *3D Grid Data* folder, *grid (1)*, the *Material Sets* folder, and the *Simulation* folder.
5. Click on any of the material sets labeled **Simulation X**. You may wish to use the up and down arrow keys on your keyboard to cycle through the material sets.

6 The MODFLOW Model Data

Most of the MODFLOW data for our model (boundary conditions, well pumping rate, top and bottom elevations, etc.) has already been entered. However, we will review the MODFLOW data that are somewhat more unique to this type of simulation.

1. Select the *MODFLOW* | *LPF - Layer Property Flow* command.

At the top of the dialog, notice that the *Use material ids* option is selected for the *Layer property entry method*. This means that we will not enter an array of K (hydraulic conductivity) values as is normally the case with MODFLOW. Rather, we will use material ids to define the K values.

2. Select the *Material IDs* button.

This dialog illustrates the material IDs assigned to cells. These material IDs are inherited from the active material set generated by T-PROGS.

3. Select the *OK* button to exit the *Material IDs* dialog.
4. Select the *Material Properties* button.

This dialog is used to assign aquifer properties, including hydraulic conductivity, to each of the materials used by the model. Notice that a value has been assigned for Horizontal k and Vertical k for each material. Since this is a one layer model only the hydraulic conductivity value will be used. When the MODFLOW model is saved to disk, GMS uses the array of material IDs and the list of material properties to automatically generate the array of K values required by MODFLOW.

5. Select the *OK* button twice to exit both dialogs.

7 Selecting the Stochastic Option

Before running MODFLOW, we need to turn on the appropriate stochastic simulation options. First, we will select the stochastic run option:

1. Select the *MODFLOW | Global Options* command.
2. In the *Run Options* section of the dialog, select the *Stochastic Simulation* option.
3. Choose *OK* to exit the dialog.

Next, we need to specify that we will be using the material set method (as opposed to parameter randomization) in our stochastic simulation. When we choose the material set option, we must also specify which group (folder) of material sets we wish to use. In our case, we only have one group called *Simulation*.

4. Select the *MODFLOW | Stochastic* command.
5. Select the *Material sets* method.
6. Verify that **Simulation** shows up in the combo box below the *Material sets* option.
7. Choose *OK* to exit the dialog.

8 Saving the Project and Running MODFLOW

We are now ready to save the project and run MODFLOW in stochastic mode.

1. Select the *File | Save As* command.
2. Save the project with the name **matsto.gpr**.
3. Select the *MODFLOW | Run MODFLOW* command.

MODFLOW is now running in stochastic mode. As each model run finishes, the spreadsheet at the top will indicate whether or not the run converged.

9 Reading in and Viewing the MODFLOW Solutions

Once all the MODFLOW runs are completed, you can read in the solutions.

1. Make sure the *Read solution on exit* toggle is checked and select the *Close* button.

2. Select *OK* at the prompt to read in all converged solutions.

You should see a new folder named **matsto (MODFLOW)(STO)** appear in the *Project Explorer*. You may wish to expand this folder and view the individual solutions. You will notice that the contours vary greatly depending on the distribution of materials. The head loss occurs primarily in the clay zones. As you view a particular solution, notice that the material set is updated to correspond to the material set used to generate that particular solution.

10 Displaying Pathlines

Before performing the probabilistic capture zone analysis, we will first view the capture zones corresponding to individual solutions.

1. Select the *MODPATH | Generate Particles at Wells* command.

2. Select the *OK* button to accept the default options.

You should see a set of pathlines appear. As you click on different MODFLOW solutions in the *Project Explorer*, the pathlines will be automatically updated. Notice how dramatically the capture zone changes from one solution to the next.

11 Probabilistic Capture Zone Analysis

Now that we have a MODFLOW stochastic solution set, we can perform a probabilistic capture zone analysis for the wells in our model. A probabilistic capture zone analysis is performed by placing one or more particles in each cell and tracking the particles forward in time using MODPATH to see if they reach a well. If any of the particles from a cell reach the well, a counter for the cell is updated. After running MODPATH on each of the MODFLOW solutions, the percentage of particles from a particular cell that are eventually captured by the well is computed and saved as the capture probability for the cell. The capture probability data set can then be contoured.

1. Right-click on *matsto (MODFLOW)(STO)* in the *Project Explorer*.

2. Select the *Risk Analysis* command from the pop-up menu. This brings up the *Risk Analysis Wizard*.

3. Verify that **MODFLOW** is selected in the list box and select the *Probabilistic capture zone analysis* option.

4. Select the *Next* button.

You should now see the next page in the *Risk Analysis Wizard*. We can use this page to specify options for the position and number of particles for each cell. In the *Particle starting locations* area, we can choose to place particles on the water table or at the cell centers. We will use the default and place particles at the water table. We can also specify the tracking duration, although for this tutorial, we will be tracking the particles until they terminate ("To end").

5. Select the *Finish* button.

At this point, the wizard should go away and a progress bar should appear at the bottom. When the computations are finished, a new data set, *well*, will be added to the *matsto (MODFLOW)(STO)* folder. This data set contains the probability that any particles placed at the water table will reach the well. The best way to view this data set is to turn on color filled contours.

6. Select *Contour Options* from the main toolbar.
7. Verify that the *Legend* option is turned on.
8. Change the *Contour Method* to **Color Fill**.
9. Select the *Color Ramp* button.
10. Change the *Palette method* to **Hue ramp**.
11. Click the *Reverse* button to put red at the max and blue at the min.
12. Click *OK* twice to exit both dialogs.

You should now see a zone of probability extending from the well. You may want to uncheck the *Particle Sets* folder in the *Project Explorer* to make it easier to see the contours.

12 Conclusion

This concludes the Stochastic Modeling – Indicator Simulations tutorial. Here are the things that you should have learned in this tutorial:

- GMS supports two types of stochastic approaches: parameter randomization and indicator simulations
- The *Risk Analysis Wizard* can be used to do a probabilistic capture zone analysis.

