GMS 9.2 Tutorial

Geostatistics – 3D

Learn the various 3D interpolation methods available in GMS
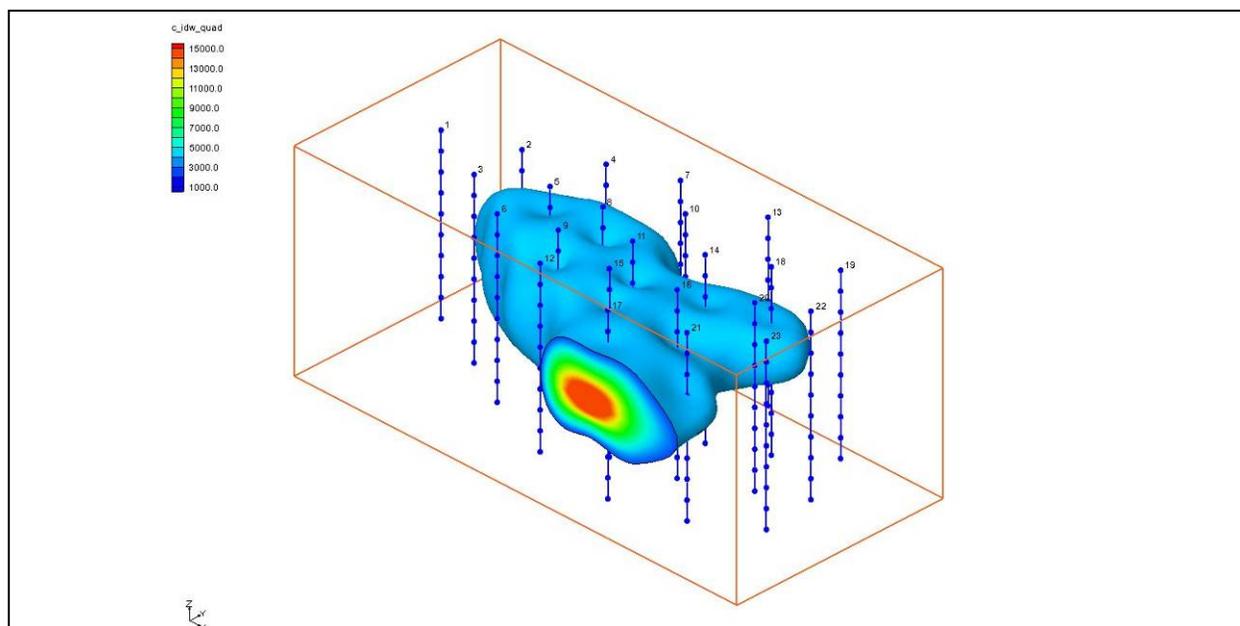
Objectives

Explore the various 3D interpolation algorithms available in GMS including IDW and kriging. Visualize the results of interpolation though cross sections and iso-surfaces.

Prerequisite Tutorials	Required Components	Time
• Geostatistics 2D	• Geostatistics • Grid	• 30-60 minutes

AQUAVEO™

1   Contents

2   Introduction

Three-dimensional geostatistics (interpolation) can be performed in GMS using the *3D Scatter Point* module. The module is used to interpolate from sets of 3D scatter points to 3D meshes and 3D grids. Several interpolation schemes are supported, including kriging. Interpolation is useful for defining initial conditions for 3D ground water models or for 3D site characterization.

The tools for manipulating 3D scatter point sets and the interpolation schemes supported in GMS are described in this tutorial.

2.1   Outline

This is what you will do:

1. Import a scatter point set.

2. Create a bounding grid.

---

3. Create iso-surfaces by interpolating scatter points to a 3D grid using different interpolation methods.

4. Create cross sections.

5. Make a moving cross section animation.

6. Make a moving iso-surface animation.

3   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.

4   Importing a Scatter Point Set

To begin the tutorial, we will import a 3D scatter point set. A 3D scatter point set is similar to a 2D scatter point set except that each point has a z coordinate in addition to xy coordinates. As with the 2D scatter point set, one or more scalar data sets can be associated with each scatter point set representing values such as contaminant concentration, porosity, hydraulic conductivity, etc. The 3D scatter point set we will import and use with this tutorial has previously been entered into a text file using a spreadsheet. The file was then imported to GMS using the *Import Wizard* (refer to the *Geostatistics - 2D* tutorial for details on using the *Import Wizard*). The project was then saved.

To read the project:

1. Select the *Open* button .

2. Open the directory entitled **Tutorials\geos3d**.

3. Open the file named **tank.gpr**.

4. Select the *Oblique View* button .

A set of points should appear on the screen. Notice that the points are arranged in vertical columns. This hypothetical set of points is meant to represent a set of measurements of contaminant concentration in the vicinity of a leaky underground storage tank. Each column of points corresponds to a borehole or the path of a penetrometer along which concentrations were measured at uniform intervals. The goal of the tutorial is to use the tools for 3D geostatistics in GMS to interpolate from the scatter points to a grid and generate a graphical representation of the plume.

5   Displaying Data Colors

Next, we will change the display options so that the color of each point is representative of the concentration at the point.

1. Select the *Display Options* button .

2. In the 3D Scatter Data Section turn on the *Contours* option.

3. Select the *Options* button.

4. Turn on the *Legend* option.

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

Notice that most of the values are zero. The nonzero values are all at about the same depth in the holes. This pattern is fairly common when dealing with light non-aqueous phase liquids (LNAPLs) which form a pancake shaped plume and float on the water table.

6   Z Magnification

Next, we will magnify the z coordinate so that the vertical variation in the data is more apparent.

1. Select the *Display Options* button .

2. Enter a value of **2.0** for the Z magnification.

3. Select the *OK* button.

4. Select the *Frame* button .

7   Creating a Bounding Grid

To generate a graphical representation of the contaminant plume, we must first create a grid that bounds the scatter point set. We will then interpolate the data from the scatter points to the grid nodes. The grid will then be used to generate iso-surfaces.

To create the grid:

1. In the *Project Explorer*, right-click on the **tank.sp3** scatter point set and select the *Bounding 3D Grid* menu command.

2. Notice that the x, y, and z dimensions of the grid are already defined. The default values shown in the dialog cause the grid to extend beyond the scatter

points by 10% on each side. Also, default values have also been entered for the number of cells in each direction. We will leave the default values.

3. Check to ensure that the default grid type is *Mesh Centered*.

Two types of grids are supported in GMS: cell-centered and mesh-centered. While cell-centered is appropriate for groundwater models (MODFLOW), the mesh-centered approach is more appropriate when the grid will be used solely for interpolation.

4. Select the *OK* button.

5. Select the *Frame* button .

A grid should appear on the screen that just encompasses the scatter point set.

8   Simple IDW Interpolation

The next step is to select an interpolation scheme. First, we will use the inverse distance weighted interpolation scheme (IDW).

1. Select the *Interpolation | Interpolation Options* menu command.

2. Select the *Inverse distance weighted* option.

3. Select the *Options* button to the right of the *Inverse distance weighted* option.

4. In the *Nodal function* section at the top of the dialog, select the *Constant* (Shepard's method) option.

5. In the section entitled *Computation of interpolation weights,* select the *Use subset of points* option.

6. Select the *Subset…* button in the *Computation of interpolation weights* section.

7. Select the *Use nearest ___ points* option and enter **64** for the number of points.

8. Select the *OK* button to exit the *Subset Definition* dialog.

9. Select the OK button to exit the *IDW Interpolation Options* dialog.

10. Select the *OK* button to exit the *Interpolation | Interpolate Options* dialog.

To interpolate to the grid:

11. Select the *Interpolation 3D Grid* menu command.

12. Select the *OK* button.

9   Displaying Iso-surfaces

Now that we have interpolated to the nodes of the 3D grid there are several ways to visualize the contaminant plume. One of the most effective ways is to use iso-surfaces. Iso-surfaces are the three-dimensional equivalent of contour lines. An iso-surface represents a surface of a constant value (contaminant concentration in this case). To define and display iso-surfaces:

1. Select the *Display Options* button .

2. Select the *3D Grid Data* option .

3. Turn off the *Cell edges* option, and turn on the *Grid shell,* and *Iso-surfaces* options.

4. Select the *Options* button to the right of the *Iso-Surfaces* option.

5. On the first row, enter **3000.0** for the *Upper Value*.

6. On the second row, turn on the *Fill between* option.

7. Turn on the *Iso-surface faces* option.

8. Select the *OK* button to exit the *Iso-Surface Options* dialog.

9. Select the *OK* button to exit the *Display Options* dialog.

You should now see the iso-surface.

10   Interior Edge Removal

A series of edges are draped over the iso-surface plot. These edges represent the intersection of the iso-surface with the grid cells. The edges are displayed to help the user visualize the spatial variation or relief in the iso-surface. However, it is sometimes useful to inhibit the display of the edges in some areas. For example, in the regions where the plume intersects the grid the iso-surface is flat. We will turn off the display of the edges in this area since they provide little benefit.

1. Select the *3D Grid Data* folder in the *Project Explorer*.

2. Select the *Grid | Iso-Surface Option* menu command.

3. Select the *Interior edge removal* option. This removes the edges between adjacent planar facets that are coplanar.

4. Select the *OK* button.

11   Specified Range

You may have noticed that the shell of the iso-surface is all one color, but the interior of the iso-surface (where the iso-surface intersects the boundary of the grid) varies in color according to the contaminant concentration. We can change the display options so that the color variation in this region is more distinct.

1. Select the *Grid | Iso-surface Options* menu command.

2. Select the *Contour specified range* option.

3. Enter **3000** for the *Minimum value*.

4. Enter **9000** for the *Maximum value*.

5. Select the *OK* button.

12   Using the Vertical Anisotropy Option

The scatter points we are using were obtained along vertical traces. In such cases, the distances between scatter points along the vertical traces are significantly smaller than the distances between scatter points along the horizontal plane. This disparity in scaling causes clustering and can be a source of poor results in some interpolation methods.

The effects of clustering along vertical traces can be minimized using the *Vertical Anisotropy* option in the *Interpolation Options* dialog. The z coordinate of each of the scatter points is multiplied by the vertical anisotropy parameter prior to interpolation. Thus, if the vertical anisotropy parameter is greater than 1.0, scatter points along the same vertical axis appear farther apart than they really are and scatter points in the same horizontal plane appear closer than they really are. As a result, points in the same horizontal plane are given a higher relative weight than points along the z axis. This can result in improved accuracy, especially in cases where the horizontal correlation between scatter points is expected to be greater than the vertical correlation (which is typically the case due to horizontal layering of soils or due to spreading of the plume on the top of the water table).

To change the vertical anisotropy:

1. In the Project Explorer select the *3D Scatter* Data folder .

2. Select the *Interpolation | Interpolation Options* menu command.

3. Change the *Vertical anisotropy* value to **0.4**.

4. Select the *OK* button.

5. Select the *Interpolation | Interpolate → 3D Grid* menu command.

6. Enter **c_idw_const2** for the new data set name.

7.   Select the *OK* button.

As can be seen, there is now much more correlation in the horizontal direction.

13   IDW Interpolation With Gradient Planes

As discussed in the *2D Geostatistics* tutorial, *IDW* interpolation can often be improved by defining higher order nodal functions at the scatter points. The same is true in three dimensions. Next, we will try *IDW* interpolation with gradient plane nodal functions.

1.   Select the *Interpolation | Interpolation Options* menu command.

2.   Select the *Options* button to the right of the *Inverse distance weighted* option.

3.   In the *Nodal function* section at the top of the dialog, select the *Gradient plane* option.

4.   Select the *OK* button to exit the IDW Interpolation Options dialog.

5.   Select the OK button to exit the Interpolation Options dialog.

To interpolate to the grid:

6.   Select the *Interpolation | Interpolate $\rightarrow$ 3D Grid* menu command.

7.   Select the *OK* button.

14   IDW Interpolation With Quadratic Functions

Next, we will try *IDW* interpolation with quadratic nodal functions.

1.   Select the *Interpolation | Interpolation Options* menu command.

2.   Select the *Options* button to the right of the *Inverse distance weighted* option.

3.   In the *Nodal function* section at the top of the dialog, select the *Quadratic* option.

4.   In the section entitled Computation of nodal function coefficients, select the Use all points option.

5.   Select the *OK* button to exit the IDW Interpolation Options dialog.

6.   Select the OK button to exit the Interpolation Options dialog.

To interpolate to the grid:

7.   Select the *Interpolation | Interpolate $\rightarrow$ 3D Grid* menu command.

8. Select the *OK* button.

15   Other Interpolation Schemes

Two other 3D interpolation schemes, natural neighbor interpolation and kriging, are supported in GMS. However, these schemes will not be reviewed in this tutorial. You are encouraged to experiment with these techniques at your convenience.

16   Viewing the Plume With a Cross Section

While iso-surfaces are effective for displaying contaminant plumes, it is often useful to use color-shaded cross sections to illustrate the variation in the contaminant concentration. Next, we will cut a horizontal cross section through the center of the plume.

1. In the *Project Explorer* select the *3D Grid Data* folder.

2. Select the *Side View* button.

3. Select the *Create Cross Section* tool.

4. Cut a horizontal cross section through the grid by clicking to the left of the grid, moving the cursor to the right of the grid, and double clicking. Cut the cross section through the middle of the iso-surface.

5. Select the *Oblique View* button.

Before we examine the cross section, we will turn off the display of the iso-surfaces.

6. Select the *Display Options* button.

7. Turn off the *Iso-surfaces* option.

8. Select the *OK* button.

Next, we will set up the display options for the cross-section.

9. Select the *Display Options* menu command and select Cross Sections.

10. Turn on the *Interior edge removal* option.

11. Turn on the *Contours* option.

12. Select the *OK* button.

Finally, we will reset the *Contour* options.

13. Select the *Display Options* menu command and select Cross Sections.

14. Select the *Options* button to the right of the *Contours* option.

15. For the *Contour method*, select the **Color Fill** option.

16. Select the *OK* button for both dialogs.

17   Using the Truncation Option

Notice the range of contaminant concentration values shown in the color legend at the upper left corner of the *Graphics Window*. A large percentage of the values are negative. This occurs due to the fact that a higher order nodal function was used. Both the quadratic and the gradient plane nodal functions infer trends in the data and try to preserve those trends. In some regions of the grid, the values at the scatter points are decreasing as you move away from the center of the plume. This decreasing trend is preserved by the interpolation scheme and the interpolated values approach zero and eventually become negative in some areas. However, a negative concentration does not make sense. This problem can be avoided by turning on the *Truncate values* option in the *Interpolation Options* dialog. This option can be used to force all negative values to have a value of zero.

1. In the Project Explorer select the *3D Scatter* Data folder .

2. Select the *Interpolation | Interpolation Options* menu command.

3. Turn on the *Truncate values* option.

4. Select the Truncate to min/max of data set option.

5. Select the *OK* button.

To interpolate to the grid:

6. Select the *Interpolation → 3D Grid* menu command.

7. Enter **c_idw_quad_trunc** for the name of the new data set.

8. Select the *OK* button.

Notice that the minimum value listed in the color legend is zero.

18   Setting up a Moving Cross Section Animation

It is possible to create several cross sections at different locations in the grid to illustrate the spatial variation of the plume. This process can be automated using the *Animation* utility in GMS. An animation can be generated showing a color shaded cross section moving through the grid.

18.1   Display Options

Before setting up the animation, we will first delete the existing cross section, turn off the color legend, and reset the contour range.

1. In the *Project Explorer* select the *3D Grid Data* folder .

2. Select the *Select Cross Sections* tool .

3. Select the cross section by clicking on it.

4. Select the *Edit | Delete* menu command.

5. Select the *Grid | Iso-surface Options* menu command.

6. Select the *Contour* s*pecified range* option.

7. Enter **1000.0** for the *Minimum value*.

8. Enter **15000.0** for the *Maximum value*.

9. Select the *OK* button.

18.2   Setting up the Animation

To set up the animation:

1. Select the *Display | Animate* menu command.

2. Turn on the *Cross sections / Iso-surfaces* option and click *Next*.

3. Turn on the *Animate cutting plane over specified XYZ range* option.

4. Turn on the Z *cutting plane*.

5. Select the *Finish* button.

18.3   Playing Back the Animation

You should see some images appear on the screen. These are the frames of the animation which are being generated. Once they are all generated, they are played back at a high speed.

1. After viewing the animation, select the *Stop* ■ button to stop the animation.

2. When you are finished, close the window and return to GMS.

19   Setting up a Moving Iso-Surface Animation

Another effective way to visualize the plume model is to generate an animation showing a series of iso-surfaces corresponding to different iso-values.

To set up the animation:

1. Select the *Display | Animate* menu command.

2. Turn on the Cross sections / Iso-surfaces command and click Next.

3. Turn off the *Animate cutting plane over XYZ range* option.

4. Turn on the *Animate iso-surface over specified data range* option.

5. Enter **1000.0** for the *Begin value*.

6. Enter **15000.0** for the *End value*.

7. Select the *Cap above* option.

8. Select the *Display values* option.

9. Select the *Finish* button.

10. After viewing the animation, select the *Stop* ■ button to stop the animation.

11. When you are finished, close the window and return to GMS.

20   Conclusion

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

- There are several 3D interpolation algorithms available in GMS.

- Mesh-centered grids are better than cell-centered grids if you are just doing interpolation and not using MODFLOW.

- Iso-surfaces can be used to visualize the results of an interpolation.

- Vertical anisotropy can be used to help overcome the problem of grouping that is common with data collected from boreholes.