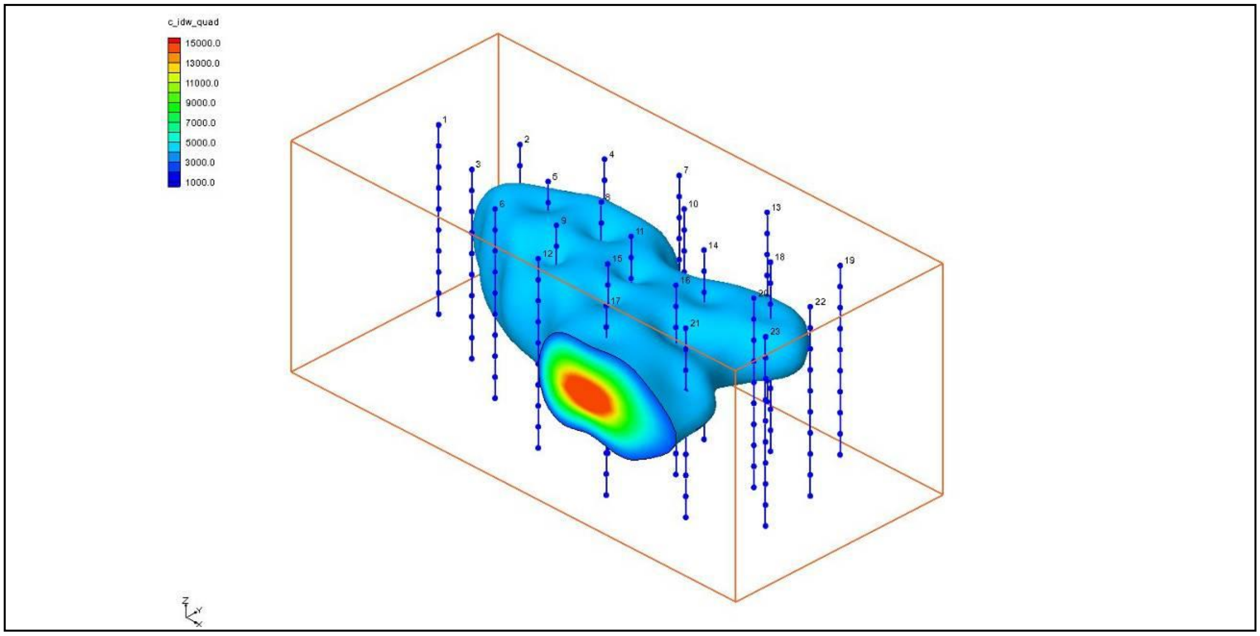
GMS

v. 10.0

# *GMS 10.0 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

- Geostatistics—2D

## Required Components

- Geostatistics
- Grid Module

## Time

- 30-45 minutes

AQUAVEO™

# 1 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.

This tutorial describes the tools for manipulating 3D scatter point sets and the interpolation schemes supported in GMS.

## 1.1 Outline

These are the steps of this tutorial:

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.

# 2 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.

# 3 Importing a Scatter Point Set

To begin the tutorial, it is necessary to 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 datasets can be associated with each scatter point set representing values such as contaminant concentration, porosity, hydraulic conductivity, etc.

The 3D scatter point set that will be imported and used 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 in the project, do as follows:

1. Select the **Open** button.
2. Open the directory entitled *Tutorials\geos3d*.
3. For *Files of type*, ensure that "All Files (*.*)" is selected.
4. Select the file named "tank.gpr."
5. Press the **Open** button.
6. 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.

# 4 Displaying Data Colors

Next, it is necessary to change the display options so that the color of each point is representative of the concentration at that point.

1. Select the **Display Options** button.
2. When the *Display Options* dialog appears, choose the 3D Scatter Data item in the list on the left, and turn on the *Contours* option.
3. Select the **Options** button to the right of the *Contours* option.
4. In the *Dataset Contour Options* dialog, turn on the *Legend* option.
5. Select the **OK** button to exit the *Dataset Contour Options* dialog.
6. Select the **OK** button to exit the *Display Options* dialog.

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) that form a pancake-shaped plume and float on the water table.

# 5 Z Magnification

Next, it is necessary to 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.

# 6 Creating a Bounding Grid

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

Do the following 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. The *Create Finite Difference Grid* dialog will appear.

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.

3. Leave the default values alone.
4. 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.

5. Select the **OK** button.
6. Select the **Frame**  button.

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

## 7 Simple IDW Interpolation

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

1. Select the *Interpolation* | **Interpolation Options** menu command.
2. In the *3D Interpolation Options* dialog, select the *Inverse distance weighted* option.
3. Select the **Options** button to the right of the *Inverse distance weighted* option.
4. The *3D IDW Interpolation Options* dialog will appear.
5. In the *Nodal function* section at the top of the dialog, select the *Constant* (Shepard's method) option.
6. In the section entitled *Computation of interpolation weights,* select the *Use subset of points* option.
7. Select the **Subset**... button in the *Computation of interpolation weights* section.
8. In the *Subset Definition* dialog, select the *Use nearest ___ points* option and enter "64" for the number of points.
9. Select the **OK** button to exit the *Subset Definition* dialog.
10. Select the **OK** button to exit the *3D IDW Interpolation Options* dialog.

11. Select the **OK** button to exit the *3D Interpolation Options* dialog.

To interpolate to the grid, do the following:

12. Select the *Interpolation* | **Interpolation → 3D Grid** menu command.
13. Select the **OK** button.

# 8 Displaying Iso-surfaces

Now that the user has 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, do the following:

1. Select the **Display Options** button.
2. Select the *3D Grid Data* option in the list on the left.
3. In the *Display Options* dialog, turn off the *Cell edges* option.
4. Turn on the *Grid shell* and *Iso-surfaces* options.
5. Select the **Options** button to the right of the *Iso-Surfaces* option.
6. In the *Iso-surface Options* dialog on the first row, enter "3000.0" for the *Upper Value*.
7. On the second row, turn on the *Fill between* option.
8. Turn on the *Iso-surface faces* option.
9. Select the **OK** button to exit the *Iso-Surface Options* dialog.
10. Select the **OK** button to exit the *Display Options* dialog.

The user should now see the iso-surface.

# 9 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. It is advisable to 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. In the *Iso-surface Options* dialog, select the *Interior edge removal* option. This removes the edges between adjacent planar facets that are coplanar.
4. Select the **OK** button.

# 10 Specified Range

The user 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. It is possible to 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. In the *Iso-surface Options* dialog, 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.

# 11 Using the Vertical Anisotropy Option

The scatter points being used 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. In the *3D Interpolation Options* dialog, change the *Vertical anisotropy* value to “0.4.”
4. Select the **OK** button.
5. Select the *Interpolation* | **Interpolate → 3D Grid** menu command.
6. In the *Interpolate →Object* dialog, enter “c_idw_const2” for the *Interpolated dataset name.*
7. Select the **OK** button.

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

## 12 IDW Interpolation with Gradient Planes

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

1. Select the *Interpolation* | **Interpolation Options** menu command.
2. In the *3D Interpolation Options* dialog, select the *Options* button to the right of the *Inverse distance weighted* option.
3. In *3D IDW Interpolation Options* dialog, find the *Nodal function* section at the top of the dialog and select the *Gradient plane* option.
4. Select the **OK** button to exit the *3D IDW Interpolation Options* dialog.
5. Select the ***OK*** button to exit the *3D Interpolation Options* dialog.

To interpolate to the grid, do as follows:

6. Select the *Interpolation* | **Interpolate → 3D Grid** menu command.
7. In the *Interpolate →Object* dialog, select the **OK** button.

## 13 IDW Interpolation with Quadratic Functions

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

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

2. In the *3D Interpolation Options* dialog, select the **Options** button to the right of the *Inverse distance weighted* option.

3. In the *3D IDW Interpolation Options* dialog, find the *Nodal function* section at the top of the dialog and 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 *3D IDW Interpolation Options* dialog.

6. Select the **OK** button to exit the *3D Interpolation Options* dialog.

To interpolate to the grid, do the following:

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

8. In the *Interpolate → Object* dialog, select the **OK** button.

# 14 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. Users are encouraged to experiment with these techniques when convenient.

# 15 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, the user 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 examining the cross section, the user should turn off the display of the iso-surfaces.

6. Select the **Display Options** button.

7. In the *Display Options* dialog, turn off the *Iso-surfaces* option.

8. Select the **OK** button.

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

9. Select the **Display Options** button.

10. When the *Display Options* dialog appears, select *Cross Sections* in the list on the left.

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

12. Turn on the *Contours* option.

13. Select the **OK** button.

Finally, the user will reset the *Contour* options.

14. Select the **Display Options** button.

15. When the *Display Options* dialog appears, be sure the *Cross Sections* is selected in the list on the left.

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

17. In the *Dataset Contour Options* dialog, select the **Color Fill** option for the *Contour method*.

18. Select the **OK** button for the *Dataset Contour Options* dialog.

19. Select the **OK** button for the *Display Options* dialog.

# 16 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 when moving away from the center of the plume. This decreasing trend is preserved by the interpolation scheme; moreover, 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. In the *3D Interpolation Options* dialog, turn on the *Truncate values* option.
4. Select the *Truncate to min/max of dataset* option.
5. Select the **OK** button.

To interpolate to the grid, do the following:

6. Select the *Interpolation* | **Interpolation → 3D Grid** menu command.
7. In the *Interpolate → Object* dialog, enter "c_idw_quad_trunc" for the *Interpolated dataset name*.
8. Select the **OK** button.

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

# 17 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.

## 17.1 Display Options

Before setting up the animation, the user 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. In the *Iso-surface Options* dialog, select the *Contour specified range* option.
7. Enter "1000.0" for the *Minimum value*.
8. Enter "15000.0" for the *Maximum value*.

9. Select the **OK** button.

## 17.2 Setting up the Animation

To set up the animation:

1. Select the *Display* | **Animate** menu command.
2. In the *Animation Wizard* dialog, 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.

## 17.3 Playing Back the Animation

The user 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. Close the window and return to GMS.

# 18 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, do as follows:

1. Select the *Display* | **Animate** menu command.
2. In the *Animation Wizard* dialog, 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. Close the window and return to GMS.

## 19 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 users are just using 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.