# GMS 6.5

## What Is GMS?

The Department of Defense **Groundwater Modeling System (GMS)** is a comprehensive graphical user environment for performing groundwater simulations. The entire GMS system consists of a graphical user interface (the GMS program) and a number of analysis codes (**MODFLOW, MT3DMS, RT3D, SEAM3D, MODPATH, MODAEM, SEEP2D, FEMWATER, WASH123D, UTCHEM**). The GMS interface was developed by the Environmental Modeling Research Laboratory of Brigham Young University in partnership with the U.S. Army Engineer Waterways Experiment Station.

GMS was designed as a comprehensive modeling environment. Several types of models are supported and facilities are provided to share information between different models and data types. Tools are provided for site characterization, model conceptualization, mesh and grid generation, geostatistics, and post-processing.

## Groundwater Flow & Transport Options

The variety of modeling options in GMS is unparalleled! Rather than being limited to one main model (such as MODFLOW) and accompanying "add-on" codes, GMS provides interfaces to a wide range of 2D or 3D models. Here is a brief overview of the options available to you:

### 2D Flow

◆ Perform fast, easy modeling with the MODAEM analytical element model integrated into GMS!

◆ 2D finite-element seeepage modeling is supported in the SEEP2D model - perfect for dams, levees, cutoff trenches, etc.

### 3D Flow

◆ 3D finite difference modeling with MODFLOW 2000 (saturated zone)

◆ 3D finite-element modeling with FEMWATER (saturated and unsaturated zone)

### Solute Transport

◆ Simple analytical transport modeling with ART3D

◆ Simple 3D transport with MT3D, MODPATH, or FEMWATER

◆ Reactive 3D transport with RT3D or SEAM3D

◆ Multi-phase reactive transport with UTCHEM

# GMS 6.5

**Unsaturated Zone Flow and Transport**

◆ Fully 3D unsaturated/saturated flow and transport modeling with FEMWATER or UTCHEM

## GIS-based Model Conceptualization

One of GMS's greatest strengths traditionally has been the conceptual model approach. This approach makes it possible to build a conceptual model in the GMS Map Module using GIS feature objects (points, arcs, and polygons). The conceptual model defines the boundary conditions, sources/sinks, and material property zones for a model. The model data can then be automatically discretized to the model grid or mesh. The conceptual model approach makes it possible to deal with large complex models in a simple and efficient manner.

The GIS Module now available in GMS has made creating conceptual models from GIS data even easier. With direct linkage to ArcGIS and almost any format of GIS data, you can access geometry and attributes faster than ever before.

Whether the GIS data is created in GMS or imported from GIS files, the method of model building remains the same. You edit the model at a GIS object level and let GMS do the hard work of grid or mesh building and parameter assignment to each element of the model.

## 3D Model Conceptualization

GMS presents new and improved tools for the creation of complex 3D stratigraphy models and the ability to translate that 3D object direclty to a finite-difference grid model or fininte-element mesh model.

The "Horizons" approach allows you to create complex solids from borehole and cross section data quickly and easily. These tools alow you to create solids with complex stratigra-phy such as pinch out zones, truncations, and outcroppings.
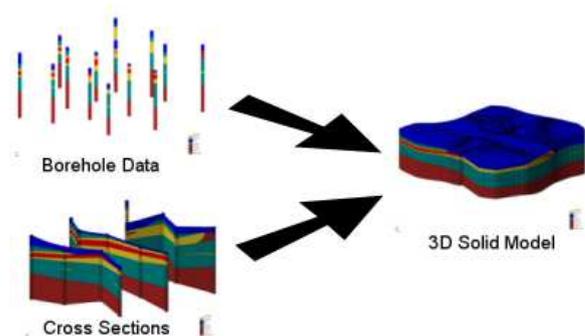


Borehole Data

Cross Sections

3D Solid Model

You can transfer the results (material properties) of a solid model direclty to a numerical model such as a MODFLOW grid or a FEMWATER mesh.  You can also direclty generate MODFLOW 2000 HUF data - GMS is the only system that allows you to do this!

2

# GMS 6.5

## Site Visualization

GMS is a powerful graphical tool for model creation and visualization of results. Models can be built using digital maps and elevation models for reference and source data. During the model building process, the graphical representation of the model allows quick review and presentation of your work. Fully 3D views, with contouring and shading, of your model allow anyone to see and understand the domain and parameters of your analysis.
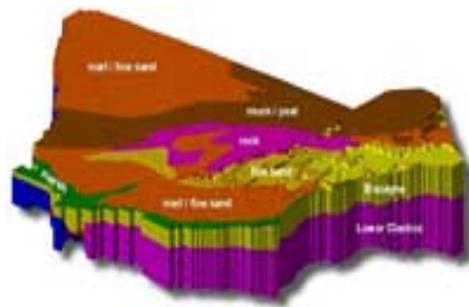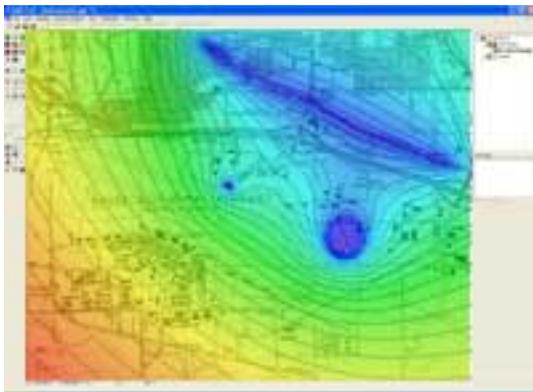


A groundwater model can be displayed in plan view or 3D oblique view, and rotated interactively. Cross-sections and fence diagrams may be cut arbitrarily anywhere in the model. Hidden surface removal, and color and light source shading can be used to generate highly photorealistic rendered images. Contours and color fringes can be used to display the variation of input data or computed results. Cross-sections and iso-surfaces can be interactively generated from 3D meshes, grids, and solids, allowing the user to quickly visualize the 3D model.
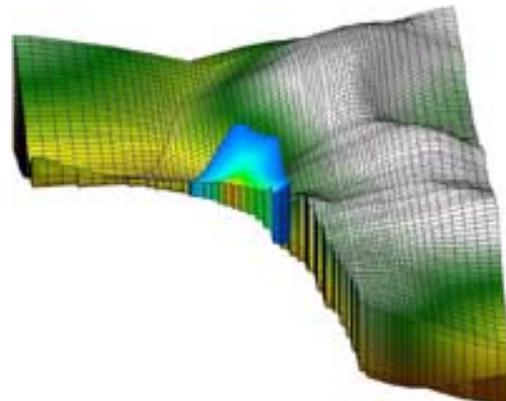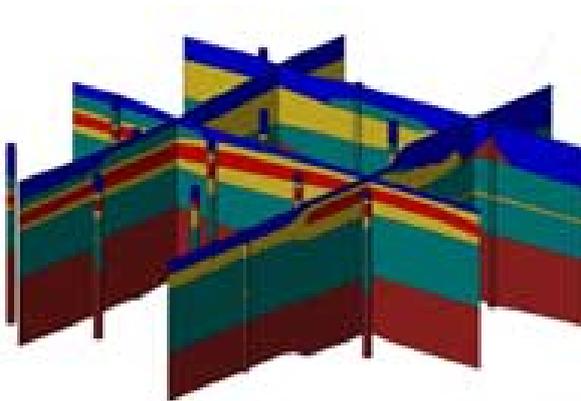


Both steady-state and transient solutions can be displayed in an animated format (as if viewing a movie) using either vector, iso-surface, color fringe, or contour animation. For example, animation of a transient solution allows the user to observe how head, drawdown, velocity, and contaminate concentration vary with time. In addition, GMS can also sweep an iso-surface through the 3D model. The minimum and maximum iso-surface values are determined from the model and the program will then linearly interpolate and display multiple iso-surfaces in rapid succession. This allows the user to quickly understand the spatial variation of a contaminant plume, for example.

# GMS 6.5

## Risk Assessment (Stochastic) Modeling

One of the most exciting features in GMS is a suite of tools for performing stochastic simulations with MODFLOW and accompanying transport models.

The Risk Analysis Wizard is a new tool associated with the stochastic modeling tools in GMS. Two types of analysis are currently supported: probabilistic threshold analysis and probabilistic capture zone delineation.  This wizard allows you to quantify the risk of a contaminant exceeding critical levels in groundwater or the risk of a capture zone including key areas at a site.  Such analysis helps determine appropriate action to be taken in design or remediation.

Two approaches are supported for setting up stochastic simulations: parameter randomization and indicator simulation.  The parameter randomization can be done using either a "Monte Carlo" or a "Latin Hypercube" approach. The indicator simulation approach randomizes the spatial distribution of the parameter zones using the T-PROGS software.   The T-PROGS software is used to perform transition probability geostatistics on borehole data. The output of the T-PROGS software is a set of N material sets on a 3D grid. Each of the material sets is conditioned to the borehole data and the materials proportions and transitions between the boreholes follows the trends observed in the borehole data. These material sets can be used for stochastic simulations with MODFLOW.
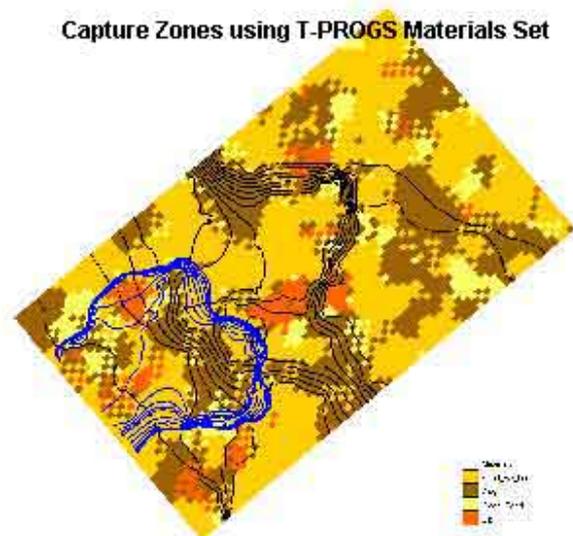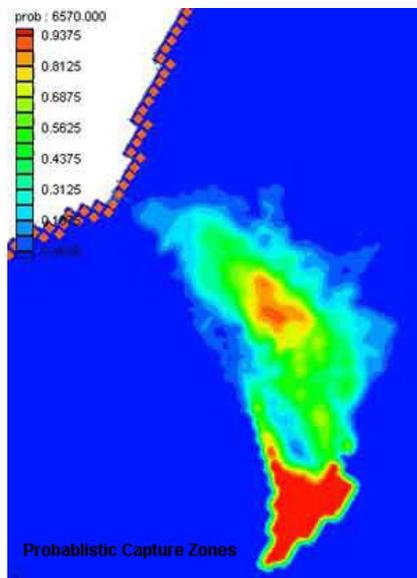
# GMS 6.5

## Automated Model Calibration

Calibration is the process of modifying the input parameters to a groundwater model until the output from the model matches an observed set of data. GMS includes a suite of tools to assist in the process of calibrating a groundwater model to point and/or flux observations. When a computed solution is imported to GMS, the point and flux residual errors are plotted on a set of calibration targets and a variety of plots can be generated showing overall calibration statistics. Most of the calibration tools can be used with any of the models in GMS.
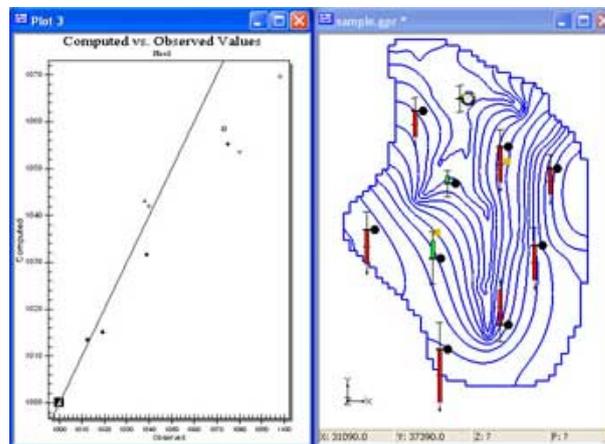


Automated parameter estimation is supported in GMS for the MODFLOW simulations using MODFLOW PES, PEST, and UCODE. These are sometimes called "inverse models". Most of the steps involved in setting up an inverse model in GMS are the same regardless of the selected inverse model. The basic process for inverse modeling is:

◆ Build a base model with MODFLOW

◆ Input observed data (point or flux data)

◆ Indicate the model input parameters that the inverse model can adjust to make the model match the observations.

◆ Let the inverse model run - it will adjust input parameters and run the MODFLOW simulation repeatedly until the best match betweeen computed data and observed data is obtained.
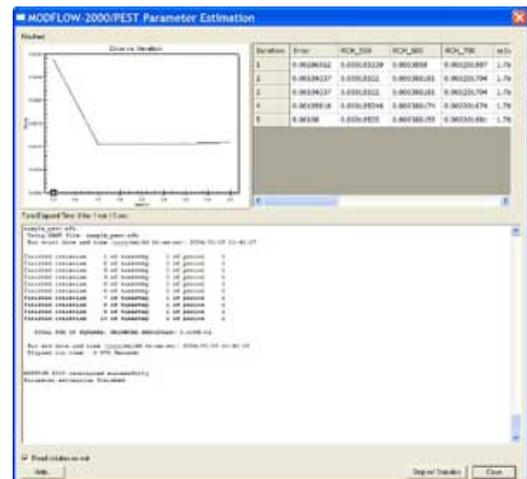


You can transfer the results (material properties) of a solid model direclty to a numerical model such as a MODFLOW grid or a FEMWATER mesh.  You can also direclty generate MODFLOW 2000 HUF data - GMS is the only system that allows you to do this!

# GMS 6.5

## GMS Supported Models

Numerical models are programs that are separate from GMS that are used to run an analysis on a model. The models can be built in GMS, and then run through the numerical model program. GMS can then read in and display the results of the analysis.  GMS also has the option of using a "model wrapper" to run the model and display real-time results of during the model simulation.

The following numerical models are currently supported in GMS. Each model is included with the GMS installation (model executable files and documentation) and is fully linked with the GMS software.

◆ **MODFLOW 2000**

GMS includes a comprehensive graphical interface to MODFLOW 2000. MODFLOW is a 3D, cell-centered, finite difference, saturated flow model developed by the USGS

◆ **MT3DMS**

Simulation of multi-species transport by advection, dispersion, and chemical reactions of dissolved constituents in groundwater systems.

◆ **SEAM3D**

A reactive transport model used to simulate complex biodegradation problems involving multiple substrates and multiple electron acceptors.

◆ **MODAEM**

Analytic element model for simple flow and transport computations

◆ **SEEP2D**

A 2D finite-element groundwater model designed to be used on profile models such as cross-sections of earth dams or levees.

◆ **MODPATH**

A particle tracking code that is used in conjunction with MODFLOW.  Particles are tracked through time assuming they are transported by advection.

◆ **RT3D**

An advanced multi-species reactive transport model developed by the Battelle Pacific Northwest National

◆ **ART3D**

A three-dimensional analytic reactive transport model developed by Dr. T. Prabhakar Clement.

◆ **FEMWATER**

A fully 3D finite-element model used to simulate density-driven coupled flow and contaminant transport in saturated and unsaturated zones.

◆ **UTCHEM**

A multi-phase flow and transport model developed by the Center for Petroleum and Geosystems Engineering at the University of Texas at Austin. UTCHEM is ideally suited for pump and treat simulations.

◆ **PEST**

UTEXAS quickly analyzes dams, levees and other slopes for the critical failure surface and factor of safety.

◆ **T-PROGS**

Used to perform transition probability geostatistics on borehole data.

◆ **UTEXAS**

A particle tracking code that is used in conjunction with MODFLOW.  Particles are tracked through time assuming they are transported by advection.

## GMS Modules

The GMS interface is separated into several modules; these modules contain tools that allow manipulation and model creation from different data types. The modules of GMS are:

◆ **Map Module**

The Map module provides a suite of tools for using GIS objects to build conceptual models, adding annotation to a plot, displaying digital background maps, and displaying CAD drawings.

Map objects are used to provide GIS capabilities within GMS. These objects include points, arcs, and polygons. Feature objects can be grouped into layers or coverages. A set of coverages can be constructed representing a conceptual model of a groundwater modeling problem. This high level representation can be used to automatically generate MODFLOW and MT3DMS numerical models. Feature objects can also be used for automated mesh generation for FEMWATER or SEEP2D numerical models.
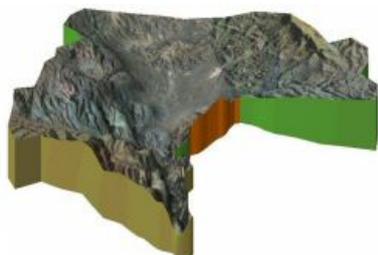


Images are scanned maps or aerial photos imported from TIFF or JPEG files. Images are displayed in the background for on-screen digitizing or model placement or simply to enhance the display of a model. Images can also be draped over surfaces and texture mapped to generate highly realistic shaded images.

The Map Module of GMS allows you to use data from many other software systems. Some of the file formats that GMS can read/write for this type of data are: **ArcGIS Shapefiles / USGS DLG files / CAD DXF files / Georeferenced or regular TIFF files / Georeferenced or regular JPEG files**

# GMS 6.5

◆ **GIS Module**

A new "GIS Module" has been added to version 5.0. This module greatly simplifies the process of importing and converting GIS data from external sources. To fully utilize the GIS module, a license of ArcGIS (formerly ArcView) version 8.0 is required. In this case, portions of ArcGIS are effectively run inside of GMS using the new ArcObjects library. This makes it possible to use the GIS module to open virtually ANY GIS database supported by ArcGIS.

Once a GIS database is opened, the data are displayed in the GMS window using the ArcGIS map rendering engine. This results in beautiful, professional looking maps that can be displayed in the background of a GMS modeling project. The user has access to all of the standard ArcGIS tools for modifying the display of the map.

Once a GIS database is imported and displayed, the user can select a portion of the map using either a simple graphical selection or an SQL query. The selected data can then be converted to standard GMS feature objects using a simple and intuitive GIS Property Mapping Wizard. The user is prompted to indicate how each of the columns in the GIS attribute data should be mapped to corresponding GMS feature object properties
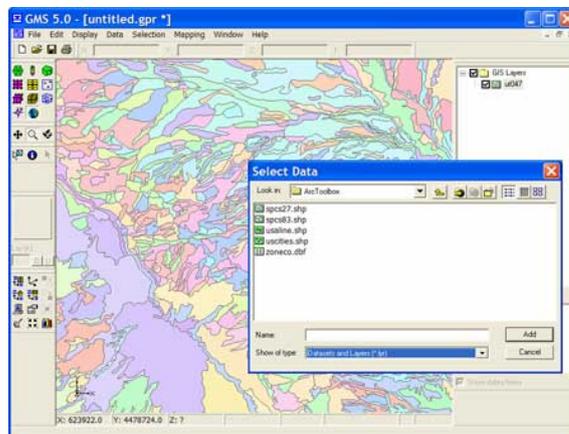
◆ **TIN Module**

The Triangulated Irregular Network (TIN) module is used for surface modeling. TINs are formed by connecting a set of XYZ points (scattered or gridded) with edges to form a network of triangles. The surface is assumed to vary in a linear fashion across each triangle. TINs can be used to represent the surface of a geologic unit or the surface defined by a mathematical function.
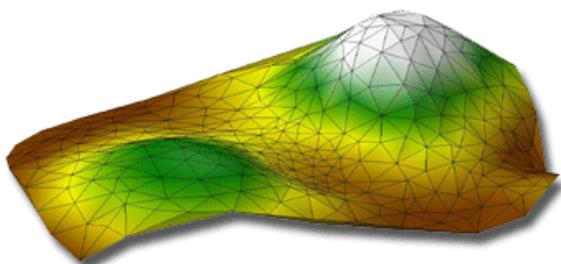
Several TINs can be modeled at once in GMS. A TIN may be created within GMS by several methods or can be imported from other systems. TINs can be used in GMS to build solid models and 3D meshes or they can be converted to other types of data such as scatter point for interpolation to grids.

# GMS 6.5

## ◆ Solids Module

The Solid module of GMS is used to construct three-dimensional models of stratigraphy using solids. Once such a model is created, cross sections can be cut anywhere on the model and the solid model can be shaded to generate realistic images. The new "Horizons Method" of constructing solids is the most advanced tool available for creating solids quickly and accurately.
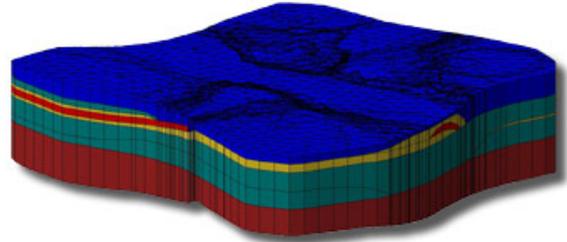


Solids are used for site characterization and visualization. Solids can also be used to define layer elevation data for MODFLOW models using the Solids -> MODFLOW command or Solids to HUF and to define a layered 3D mesh using the Solids -> Layered Mesh.

## ◆ 2D Grid Module

The 2D Grid module is used for creating and editing two-dimensional Cartesian grids. 2D grids are primarily used for surface visualization and contouring. This is accomplished by interpolating to the grid and then shading the grid. The figure below is an example of interpolating contaminant concentration data to a 2D grid and then shading the 2D grid.
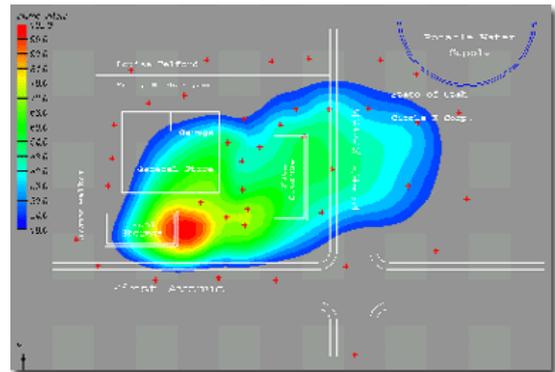


## ◆ 3D Grid Module

The 3D Grid module is used to create 3D Cartesian grids. These grids can be used for interpolation, iso-surface rendering, cross sections, and finite difference modeling.

Interfaces to the following 3D finite difference models are provided in this module. Clcik below for a more complete description of each model:

- MODFLOW
- MODPATH
- MT3DMS
- RT3D
- SEAM3D
- UTCHEM
- ART3D
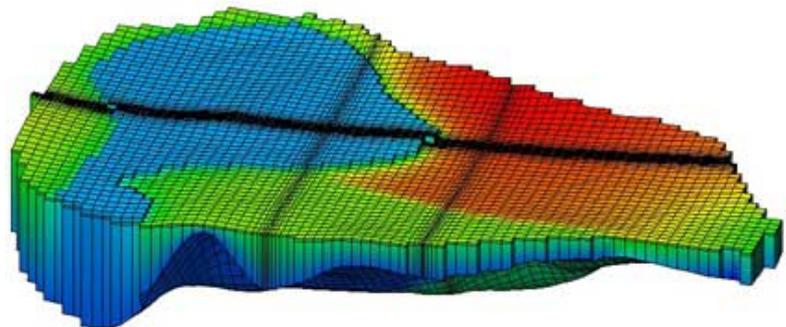
# GMS 6.5

## ◆ 2D Mesh Module

The 2D Mesh module is used to construct two-dimensional finite element meshes. Numerous tools are provided for automated mesh generation and mesh editing. 2D meshes are used for SEEP2D modeling and to aid in the construction of 3D meshes. The figures below show an example of a SEEP2D model.
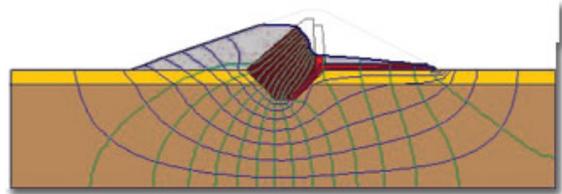


## ◆ 3D Mesh Module

The 3D Mesh module is used to construct three-dimensional finite element meshes. Numerous tools are provided for automated mesh generation and mesh editing. These meshes can be used for interpolation, iso-surface rendering, cross sections, and finite element modeling with FEMWATER
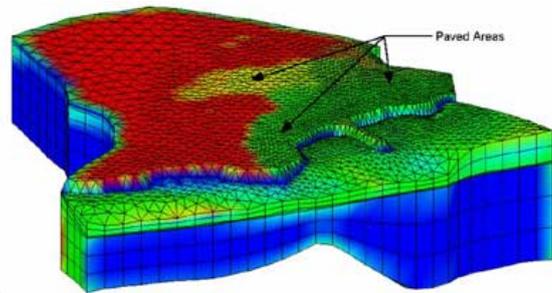


## ◆ 2D / 3D Scatter Point Module

The 2D Scatter Point module is used to interpolate from groups of 2D scattered data to other objects (meshes, grids, TINs). Several interpolation schemes are supported, including kriging. Interpolation is useful for setting up input data for analysis codes and for site characterization. The interpolation methods supported by the 3D Scatter Point module are:

- Linear
- Inverse Distance Weighted
- Clough-Tocher
- Natural Neighbor
- Kriging

The 3D Scatter Point module is used to interpolate from groups of 3D scattered data to other objects (meshes, grids, TINs). Several interpolation schemes are supported, including kriging. Interpolation is useful for setting up input data for analysis codes and for site characterization. The interpolation methods supported by the 3D Scatter Point module are:

- Linear
- Inverse Distance Weighted
- Clough-Tocher
- Natural Neighbor
- Kriging
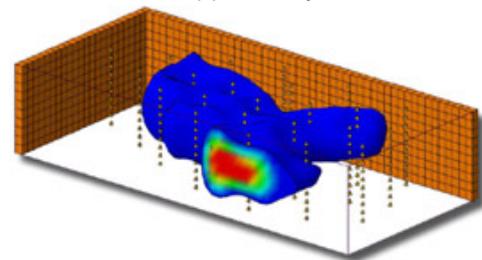


소프트팩토리
www.softfactory.kr
Tel : (+82) 010-4027-7007

SoftFactory
WATER AND ENVIRONMENT