Vertical Mapper®

Version 3.5

USER GUIDE

Pitney Bowes MapInfo

Americas:
Phone: (518) 285-6000
Fax: (518) 285-6070
Sales: (800) 327-8627
Government Sales: (800) 619-2333
Technical Support: (518) 285-7283
Technical Support Fax: (518) 285-6080
www.mapinfo.com

UK and EMEA:
Phone: 44 1753 848200
Fax: 44 1753 621140
Technical Support: 44 1753 848229
www.mapinfo.co.uk

Asia Pacific:
Phone: 61. 2.9437.6255
Fax: 61.2.9439. 1773
Technical Support: 1800 648 899
www.mapinfo.com.au

Contact information for all Pitney Bowes MapInfo offices is located at: **http://www.mapinfo.com/contactus**.

November 2008

Table of Contents

1

Introduction to Vertical Mapper

This manual provides you with the necessary information to create mapping solutions for a wide range of applications.

This chapter covers the documentation provided with Vertical Mapper™. It also details how to get technical support and how you can provide feedback on the documentation.

In this chapter:

What's New in Vertical Mapper 3.5

Smoothing a Grid

The Vertical Mapper 3.5 contains a new **Smoothing** feature that reduces the variability in a numeric grid by averaging cell values contained in a specific area. In general there are two categories of filters. The first category includes low pass (averaging) filters used for smoothing the variation of cell values found in a grid. The second category consist of edge detection or high pass filters, these are typically used for enhancing edges in grids and images. The new Vertical Mapper **Smoothing** feature uses the first category of these filters. For more information on Smoothing feature, see **Smoothing a Grid on page 108**.

Registering Vertical Mapper as MapInfo Professional Tool

The Vertical Mapper tool gets registered during installation provided MapInfo Professional 9.0 or higher version is already installed. For more information on how to register Vertical Mapper, see **Registering Vertical Mapper as MapInfo Professional Tool on page 14**

Merged all Public and Private Updates to Date

Multiple enhancements and bugs fixes were incorporated by merging private builds that were not released publicly until now.

Updated Vertical Mapper SDK

Vertical Mapper SDK is a development kit used by programmers to take advantage of Vertical Mapper capabilities. While the SDK documentation was version 2.6, the shipping product version was 3.1.1. As such, a gap existed between the SDK documentation and the released software in terms of changed and new functions. For example: documented functions in VM SDK v2.6 were incorrect. In addition, through the private builds, new functions were introduced but not documented as part of the SDK.

Enhancements made:

Vertical Mapper SDK documentation was updated and synchronized with Vertical Mapper the product. This included:

1. Corrected documented functions in the SDK.
2. Exposed new functions that were not previously available in the SDK.

MapBasics SDK support:

1. MapBasic .def files are now included with Vertical Mapper SDK v3.5.
2. MapBasic access to Vertical Mapper SDK is now a part of the standard SDK documentation.
3. Programmers can quickly start using Vertical Mapper SDK from MapBasic by leveraging the included sample applications demonstrating how to access Vertical Mapper functions from MapBasic.

Upgrade to the Vertical Mapper MFAL library

The new MFAL library that is now available in Vertical Mapper v3.5 provides the following benefits:

1. Support of MapInfo's latest TAB file format enhancements including:
   a. Very large geographic objects with tens of millions of nodes
   b. Time & Date/Time field type support
   c. All MapInfo Professional supported datums and projections

Major Bug Fixes

Over 150 bugs have been addressed in Vertical Mapper and Vertical Mapper SDK. The included release-notes document provides detailed information describing the updates that have taken place.

Registry Information Update

Registry entries for Vertical Mapper were updated from outdated company references to Pitney Bowes MapInfo. The changes were incorporated into the Installers and the code of both Vertical Mapper and Vertical Mapper SDK.

New Installer for Vertical Mapper and Vertical Mapper SDK

Both Vertical Mapper and Vertical Mapper SDK have new installers. Primary changes to the installers are:

- New Icons / Shortcuts
- Registry entry changes
- License Agreement update
- Serial number (Pitney Bowes MapInfo format)
- Demo-shield – easy CD navigation program
- A check for MapInfo Professional v9.0 or greater to be installed (Vertical Mapper product only)
- Support for MapInfo Professional workgroup install option

Vertical Mapper SDK updates:

New Vertical Mapper v3.5 SDK CD containing:

- Documentation folder – SDK documentation PDF with all available functions
- Redistributable folder – all binary files needed for redistribution of Vertical Mapper SDK
- Samples folder with – Developer include files and code samples in various programming environments

OS Support:

1. Win 2000 Professional SP4
2. Win XP Professional SP2
3. Win XP Home SP2

4. Windows XP (64-bit)
5. Windows Vista (Ultimate)

MapInfo Professional and MapInfo RunTime version support:

- V9.0; 9.0.1; V9.0.2
- 9.5; V9.5.1

Installing or Upgrading Vertical Mapper

You install Vertical Mapper in the same way you install any other conventional application for the Windows operating system.

System Requirements

These are the minimum system requirements for Vertical Mapper 3.5:

Operating Systems	Memory	Disk Space	Graphics	Monitor
Windows 2000 Professional SP 4 Windows XP Professional SP 2 Windows XP Home SP 2 Windows XP (64-bit) Windows Vista Ultimate	128 MB of RAM with a minimum of a Pentium PC	Application 103 MB Data 450 MB	16- or 24-bit Color	800x600 Display
Windows 2003 Enterprise Server SP 1 with Terminal Services/Citrix 4.0	64 MB of RAM with a minimum of a Pentium PC	Application 103 MB Data 450 MB	16- or 24-bit Color	800x600 Display

These are the recommended system requirements for Vertical Mapper 3.5:

Operating Systems	Memory	Disk Space	Graphics	Monitor
Windows 2000 Professional SP 4 Windows Vista Ultimate Windows XP Professional SP 2 Windows XP Home SP 2 Windows XP (64-bit)	256/512 MB of RAM with a minimum of a Pentium PC or better	Fast EIDE 2 or SCSI Interface with 2GB or better Data 450 MB	Mid to High 2D/3D card with 128MB or better	Greater than 1024x768 resolution or better
Windows 2003 Server SP 1 with Terminal Services/Citrix 4.0	Same, PLUS memory sufficient to support each connected user	Fast EIDE 2 or SCSI Interface with 2GB or better Data 450 MB	*Server*: Same *Client*: Choose based on resolution/speed requirements	Greater than 1024x768 resolution or better

Note **For Windows 2003 Server Users**: Keep in mind that some of the options you choose for the server can limit the options available to the client system.

Installing Vertical Mapper for the First Time

MapInfo Professional 9.0 or higher version must be installed before you can install Vertical Mapper 3.5. Before installing Vertical Mapper, you must close MapInfo Professional.

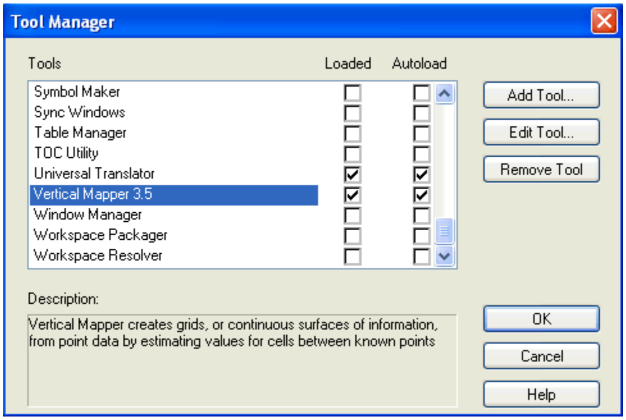
To install Vertical Mapper:

1. Insert the Vertical Mapper CD-ROM into your computer's CD-ROM drive (for example, D:\). The operating system automatically detects the Vertical Mapper autorun command and begins the installation process.

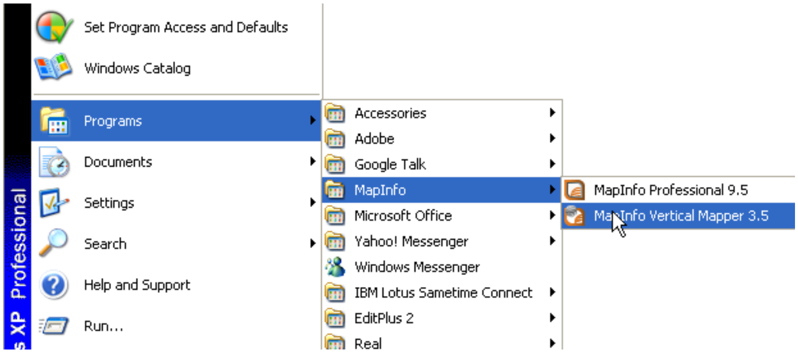
   If the Vertical Mapper Installation Wizard does not appear, use Windows Explorer to locate the **install.exe** file on the CD-ROM and double-click it to launch the wizard.
2. On the first page of the Wizard, click **Install Vertical Mapper**.
3. On the Welcome page, click **Next**.
4. Read the license agreement. If you accept the terms of the agreement, click **Yes**.
5. On the Product ID page, enter your name, your company name, and your 15 digit product ID number found on the CD case, and click **Next**.
6. On the Choose Destination Location page, choose the folder where you want to install Vertical Mapper and click **Next**.
7. On the Select Components page, choose the components you want to install and click **Next**.
8. When the installation process has finished, click **Finish**.
9. Click **Exit**.

Registering Vertical Mapper as MapInfo Professional Tool

The Vertical Mapper tool gets registered during installation provided MapInfo Professional 9.0 or higher version is already installed.



You would run Vertical Mapper as any other program by going to **Start**->**Programs** -> **MapInfo** -> **MapInfo Vertical Mapper 3.5**.



Upgrading Vertical Mapper

Vertical Mapper 3.5 does not support Upgrading.

If you already have Vertical Mapper 3.1 or earlier, please uninstall the old version and install Vertical Mapper 3.5.

1. Insert the Vertical Mapper CD-ROM into your computer. If the Vertical Mapper Installation Wizard does not appear, use Windows Explorer to locate the setup.exe file on the CD-ROM and double-click it to launch the wizard.
2. On the first page of the Wizard, click the **Install Products** button.
3. On the Install Products page, choose the **MapInfo Vertical Mapper** option, and follow the instructions of the Installer.
4. When the installation process has finished, click **Finish**.

5. Click **Exit** to exit the DemoShield application or see what is available on the CD or find contact information.

Getting Help

You can access help by choosing the Help for Vertical Mapper command from the Vertical Mapper menu. You can also access help from within a dialog box by pressing **F1**.

Vertical Mapper Documentation

Vertical Mapper comes with the following documentation.

Document	Description
Vertical Mapper User Guide	This guide explores the features of Vertical Mapper, explaining how to create grids, perform operations on spatial data that is stored in grids, and display, analyze, and export digital elevation models and other grid-based data.
Vertical Mapper Help System	Help is available in Vertical Mapper from within a dialog box when you press F1. You can also access the help from the Vertical Mapper menu.
Vertical Mapper Tutorial	When you install Vertical Mapper, you can choose to also install the Vertical Mapper tutorial and accompanying data files. They will be installed in a folder called VM\Tutorials. The Vertical Mapper Tutorial provides an overview of the basic functionality of Vertical Mapper. It consists of a series of lessons, based on practical examples of day-to-day GIS needs, ranging from simple to more complex scenarios. You can use the accompanying data files to work through the lessons. The Vertical Mapper tutorial is available only as a .pdf file. If you have installed it, you can access it from the Vertical Mapper menu.

Developing Applications with Vertical Mapper

The Vertical Mapper Software Developer’s Kit is available as a separate product. It gives you access to Vertical Mapper interpolation, modeling, contouring, and analysis functions, enabling you to build custom applications and automate repetitive operations.

Getting Technical Support

You can get technical support by email, fax, or phone.

In the Americas:	Tel 518.285.7283 Fax 518.285.6080 E-mail techsupport@mapinfo.com
In Europe/Middle East/Africa:	Tel 44.1753.848229 Fax 44.1753.621140 E-mail support-europe@mapinfo.com
In Asia-Pacific:	Tel 61.2.8925.7370 Fax 61.2.9439.1773 E-mail ozsupport@mapinfo.com

When you call for technical support, ensure that you have your product ID number and know which version of the software you are running. You can obtain this information using the About Vertical Mapper command from the Vertical Mapper menu.

Send us Your Comments

We welcome any comments you have about our documentation. Send your comments to the Documentation Manager at the following email address:

documentation@mapinfo.com

2

Understanding Grids

This chapter covers the concept of grids and explains how they are used in Vertical Mapper.

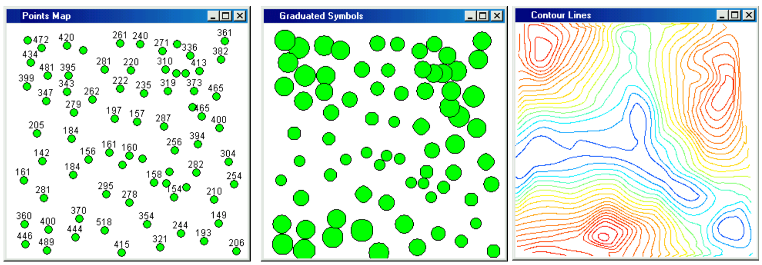
In this chapter:

Using Vertical Mapper to Display Spatial Data

Vertical Mapper brings two main benefits to MapInfo Professional:

- It provides a mapping technique for calculating and displaying the trends of data that vary continuously over geographic space.
- It provides a mechanism for sophisticated comparison and analysis of multiple map layers.

Three main object types are currently used by MapInfo Professional to represent the spatial distribution of data: regions, lines, and points. None of these objects is very well suited to representing data that varies continuously through space such as ground-level air temperature, elevation, distance from a store location, or the distribution of wealth across a city. Values for this type of data must all be collected at discrete locations, but the way they change over space is very significant. Traditional ways of indicating variation are labeling individual sample locations with a known value, creating graduated symbols at each sample site, where the size reflects the sample's value, and generating contour lines or regions depicting locations of equal value (next figure).



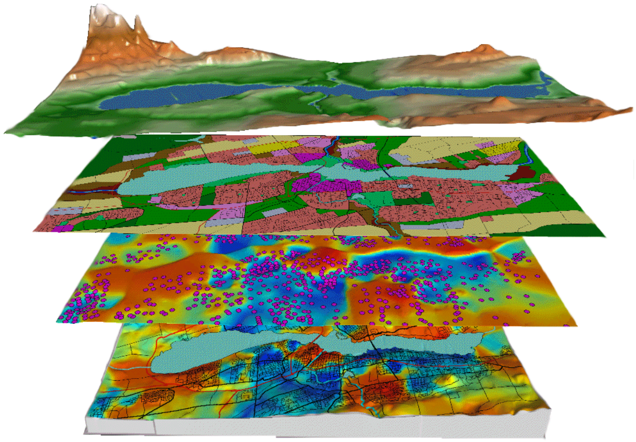
Three examples of how a traditional vector-based GIS system, such as MapInfo Professional, displays data that varies continuously.

The problem with these methods is that they do not portray how the data changes between known locations.

To address this problem, Vertical Mapper creates a type of spatial data representation for MapInfo Professional called a grid. Grids enable you to represent data as a continuous coverage. You can see how values change in space and query any location to obtain a meaningful value.

Working with Grid Files

You should be familiar with the concept of map layers when you work with Vertical Mapper. Each unique layer of information exists as a separate file that can be added as a layer in a Map window.



Various map layers covering the same geographical area can hold different types of information.

Just as each layer can be visualized above or below another layer, layers can be compared using spatial analysis functions.

MapInfo Professional works with layers of map data that are vector-based, that is, point, line and polyline, or polygon information. Points can represent soil samples or retail store locations; lines and polylines can represent roads; and polygons can represent trade areas, bodies of water, or municipal boundaries.

Vertical Mapper adds another level of data representation to MapInfo Professional, specifically, grid-based layers. Grid data is the best way to represent phenomena that vary continuously through space. Elevation, field strength, soil chemistry and income are excellent examples of properties that are distributed in constantly varying degrees through space and are best represented in grid form as map layers.

Understanding Grids

Grids represent the basic structural component for contouring, modeling, and displaying spatial data in Vertical Mapper. Grids can be considered the fourth spatial data type after regions, lines, and points.

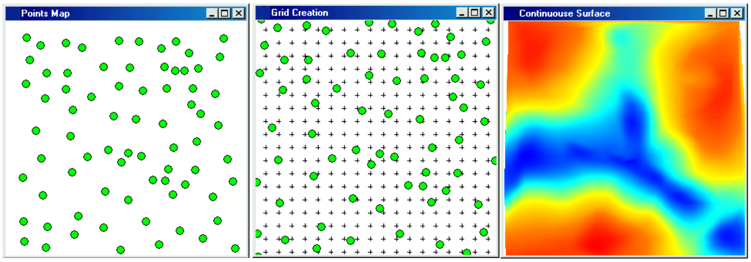
A grid can be used to effectively visualize the trends of geographic information across an area. Grids give you the power to mathematically compare and query layers of information, create new derived grids, or analyze grid layers for such unique properties as visual exposure, proximity, density, or slope.

What is a Grid?

A grid is made up of regularly spaced square cells arranged over a given area. Each cell has a node, which is a point located at its centre. Each cell can be given a value and a colour representing the value. If there are several cells between two known locations, such as two contour lines, the change in colour indicates how the values change between the locations.

The Gridding Process

Typically, the gridding process begins by overlaying an array of grid nodes over the original point file. This can be visualized as a regularly spaced point file arranged in the form of a grid (next figure). Each grid node is then attributed with an estimated value based upon the values of the surrounding points. The grid is then displayed in a Map window. The display takes the form of a raster image, where the colours reflect the estimated node values. It is because of this display component that grid-based GIS systems are called raster GIS.



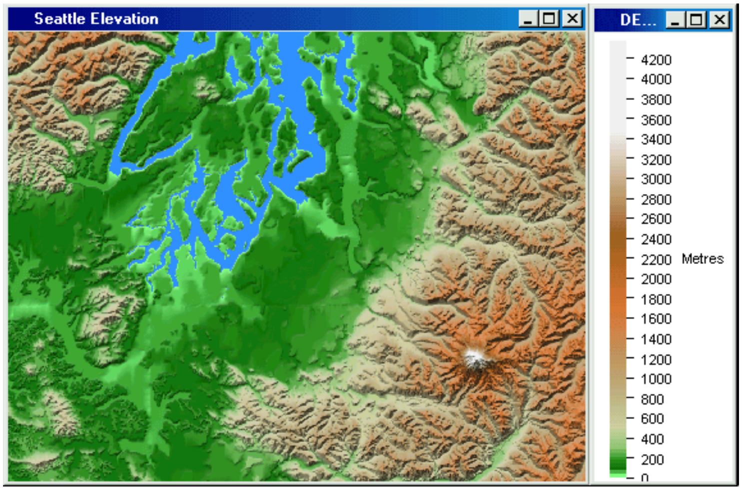
The gridding process begins by creating an array of grid nodes geographically coincident to the sample file. Each grid node is attributed with an estimated value that is displayed as a raster image in a Map window.

Grid Types

Vertical Mapper supports two types of grids: numeric grids, which have numeric attribute information, and classified grids, which have character attribute information.

Numeric Grids

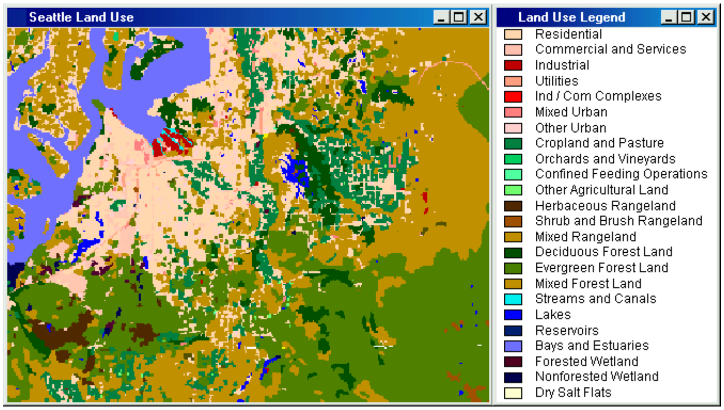
The most illustrative example of a numeric grid is a digital terrain model where each grid cell is referenced to a value measured in units of distance above sea level (next figure). Numeric grids are best used to define continuously varying surfaces of information, such as elevation, in which grid cell values are either mathematically estimated from a table of point observations or assigned real numeric values. For example, in the next figure, each grid cell was calculated (interpolated) from a table of recorded elevation points. In Vertical Mapper, numeric grid files are given the extension .grd.



An example of a numeric grid showing the continuous variation of elevation across an area.

Classified Grids

Classified grids are best used to represent information that is more commonly restricted to a defined boundary. They are used in the same way that a MapInfo Professional region is used to describe a boundary area, such as a land classification unit or a census district. In this case, the grid file does not represent information that varies continuously over space. In the next figure, a land classification grid displays each grid cell with a character attribute attached to it that describes the land type underlying it. In Vertical Mapper, classified grid files are given the extension .grc.



An example of a classified grid representing land use where each cell is referenced to a descriptive attribute.

Grid File Architecture

Every Vertical Mapper grid file (.grc or .grd) is divided into two sections. The first section, called the file header, contains several pieces of information including the following:

- Map name
- Map size (number of cells in height and width)
- Cell size
- Coordinates of first cell
- Grid Projection
- Grid value description

The second section, the body of the grid file, contains the attribute data for every cell in the map.

MapInfo Tables

In MapInfo Professional, for every table, up to five different files are created (.tab, .map, .dat, .id, and .ind). In Vertical Mapper, all the information is contained in either a .grc or a .grd file. A .tab file that points to the .grc or .grd file is also created, enabling the grid file to be opened in MapInfo Professional.

Grid Resolution

The resolution of a grid is based on the size of the cells. Vertical Mapper has square cells, so the width and height are identical. The smaller the grid, the higher the resolution, and the more detailed the information depicted. For example, what appears as a spike at 1000m resolution may be clearly discernible as part of a mountain at 1m resolution.

Appropriate resolution depends on the application. For example, for data at the world level, 1 km resolution is relatively high. For wireless planning, however, 1 km resolution is low, and 1m resolution produces much more accurate results.

Understanding Spatial Estimation Techniques

In order to represent how data changes between known values, some type of estimation must be made. There are several kinds of estimation technique that can be applied to a point file. These techniques fall into two different categories: interpolation techniques and modeling techniques. They approach the creation of a grid from entirely different perspectives, but both are mathematical construction tools designed to build grids that assign values to grid nodes from a geographically coincident point file.

Interpolation Techniques

These techniques are used to build grids that are an estimation of the same variable as the underlying points. Each new grid cell has the same unit of measure as the point value.
Vertical Mapper currently supports six interpolation techniques:

- Triangulation with Smoothing (TIN)
- Inverse Distance Weighting (IDW)

- Natural Neighbour (simple and advanced)
- Rectangular
- Kriging
- Custom Point Estimation

Modeling Techniques

These techniques create grids of derived values. For example, one of the modeling techniques included with Vertical Mapper is trade area analysis. This modeling technique uses store locations and their relative “attractiveness” to calculate grid values measuring the percent probability of customer patronage. In this case, the units of the resulting grid are different from the units of the originating point file.

Vertical Mapper currently supports two modeling techniques. The first, the Location Profiler, creates a grid measuring the average distance to point locations from anywhere within a map area. The second, trade area analysis, is based on the Huff model and calculates a grid measuring the probability of customer patronage within a trade area.

Other Grid Creation Techniques

You can also create grids using the following methods:

- Convert existing MapInfo Professional regions to a grid. This is known as vector-to-raster conversion and is given the term Region to Grid in Vertical Mapper. The process simply converts the existing region file to a grid where every grid node is given the same value as the region it falls in. Because Vertical Mapper grids can be attributed with only one piece of information, you are prompted to choose the column in the region file to attribute to the grid. In this way, you choose what type of grid will be created. If a numeric column is chosen, then a numeric grid is created. If a character column is chosen, a classified grid is created.
- Import grid files from external sources. This technique offers more flexibility with regard to the source of the information used and analyzed.
- Analyze existing grids. Each tool used to analyze an existing grid creates a new grid with the results of the analysis. There are many ways to analyze existing grids, for example, by performing a viewshed analysis, by using the Grid Calculator, by using the Grid Query tool, and by creating slope and aspect.

3

Creating Grids Using Interpolation

This chapter describes all of the basic commands associated with the creation of numeric grid files using interpolation techniques.

In this chapter:

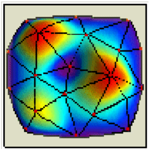
Grid Creation using Interpolation

Interpolation is the process of estimating grid values using measured observations taken from a point file. New values calculated from the original point observations form a continuous, evenly spaced grid surface that “fills in the gaps” between the non-continuous points. Many mathematical formulae can be applied to estimating or interpolating grid values from an existing point file. There is no perfect solution, and many techniques are in use. The validity of each method depends entirely upon the type of data being interpolated, and each generates a unique style of interpolation surface.

Types of Interpolation

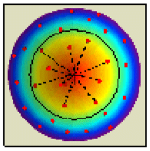
The Interpolation Wizard enables you to create grid files using six different techniques. The Natural Neighbour technique has two variants: simple and advanced.

Triangulation with Smoothing



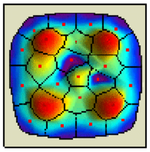
Original data points are joined by a network of lines to build a mesh of triangular faces, called a Triangular Irregular Network (TIN). These faces represent the original data surface. New grid values are then estimated according to the slope of the TIN surface at the nearest points.

Inverse Distance Weighting



Original data points lying within a prescribed radius of a new grid node are weighted according to their distance from the node and then averaged to calculate the new grid cell value.

Natural Neighbour



A network of natural neighbour regions (Voronoi diagram) is built using the original data. This creates an area of influence for each data point that is used to assign new values to overlying grid cells

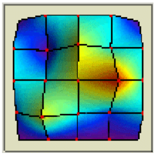
Natural Neighbour (Simple)

The Simple option offers the first-time user a two-step process for implementing the interpolation technique. Many of the controls have been pre-set to generate the most appropriate surface given the distribution of points.

Natural Neighbour (Advanced)

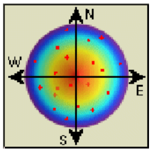
The Advanced option gives you access to a variety of controls in the Natural Neighbour interpolation technique that you can use to make subtle adjustments to the grid surface generated from a points table.

Rectangular (Bilinear)



Original data points are joined by a network of lines to build a rectilinear mesh. New grid values are then estimated using the slopes of the double linear (bilinear) framework formed by the nearest four points.

Kriging



Kriging is a geostatistical interpolation technique that considers both the distance and the degree of variation between known data points when estimating values in unknown areas. Graphing tools help you understand and model the directional (for example, north-south, east-west) trends of your data.

Custom Point Estimation

In Custom Point Estimation, grid cell values are calculated based upon a user-defined math operation and performed using the data points found within a given search radius around each cell. Math operations include sum, minimum, maximum, average, count, and median.

Choosing an Interpolation Technique

The most challenging task in creating a surface through interpolation is choosing the most appropriate technique. All interpolation techniques create gridded surfaces; however, the results may not properly represent how the data behaves through space, such as how the values change from one location to the next. For example, if an elevation surface is created from sample points taken in a mountainous area, you need to choose a technique that can simulate the severe elevation changes because this is how this type of data behaves.

It is not always easy to understand how data behaves before you start the gridding process and, therefore, it can be difficult to know what technique to use. The answers to the following questions will help you determine the most appropriate technique to use.

1. What kind of data is it, or, what do the data points represent?
   Some interpolation techniques can be automatically applied to certain data types.

Data Type	Possible Interpolation
Elevation	Triangular Irregular Network (TIN), Natural Neighbour (NN)
Soil Chemistry	Inverse Distance Weighting (IDW), Kriging

Data Type	Possible Interpolation
Demographic	NN, IDW, Kriging
Drive Test	NN

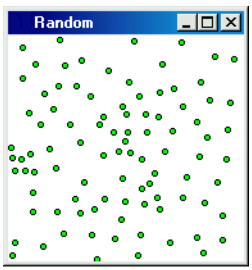
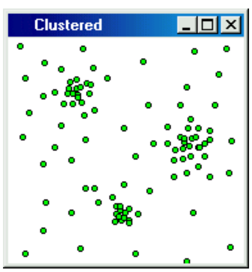
2. How accurate is the data?

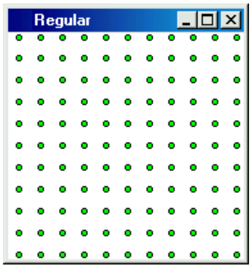
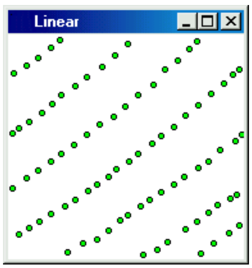
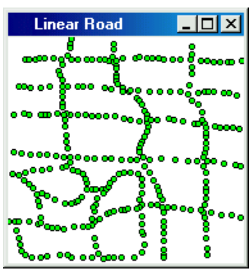
Some techniques assume that the value at every data point is an exact value and will honour it when interpolating. Other techniques assume that the value is more representative of an area.

Point Value Accuracy	Possible Interpolation Technique
Very Accurate	NN, TIN, Rectangular
Not Very Accurate	IDW, Kriging

3. What does the distribution of the points look like?

Some interpolation techniques produce more reasonable surfaces when the distribution of points is truly random. Other techniques work better with point data that is regularly distributed.

Point Distribution	Possible Interpolation Technique
	Most interpolation techniques work well with randomly scattered data points. NN, TIN, IDW, Kriging
	Highly clustered data presents problems for many interpolation techniques. NN, IDW, Kriging TIN – for slightly clustered data points

Point Distribution	Possible Interpolation Technique
	Rectangular can only handle data that is distributed in an evenly spaced pattern. Rectangular, NN, Kriging
	This type of linear pattern generally occurs when data is collected from aircraft. Samples are taken close together but flight lines are some distance apart. IDW, NN, Kriging
	This type of linear pattern generally occurs when samples are taken along roads. NN, Kriging

4. Is interpolation speed a factor?

Certain factors will influence the speed of interpolation. The smaller the cell and/or the more points in the data, the longer it takes to calculate the surface. However, some interpolation techniques are faster than others.

Interpolation Technique	Speed	Limiting Factors
TIN	Fast	None
IDW	Fast	Search and display radius size
Rectangular	Very Fast	Search radius size
NN	Slow	Point distribution
Kriging	Slow	Number of directions analyzed

5. Is it necessary to overshoot or undershoot the local minimum and maximum values?
   Overshooting and undershooting the local minimum and maximum values is generally necessary when interpolating elevation surfaces.

Over/Undershoot	Possible Interpolation Technique
Yes	TIN, NN
No	IDW, Rectangular, Kriging

Using the Interpolation Wizard to Create a Grid

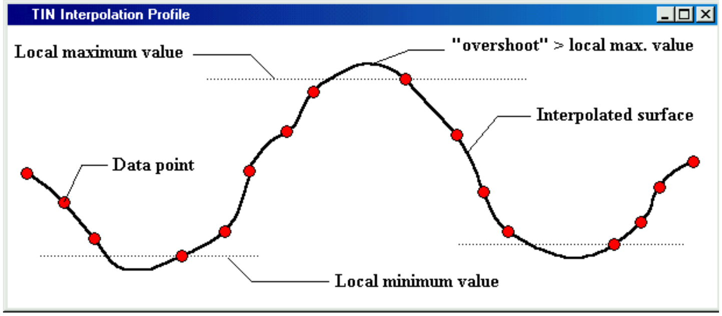
Using the Interpolation Wizard to create grid files streamlines the process of estimating grid values and enables you to make better decisions in choosing an interpolation technique that will best manage a particular data set.

1. From the Vertical Mapper menu, choose the **Create Grid > Interpolation** command.
2. In the Select Interpolation Method dialog box, choose the interpolation method you want to use.
3. Click **Next**.
4. In the Select Table and Column dialog box, click **Open Table** to add a table to the Select Table To Grid list.
5. From the Select Table To Grid list, choose the appropriate MapInfo table of points that contains the data to be gridded.
6. From the Select Column list, choose the column that contains the attribute values.
7. To use an unmapped MapInfo data file (an x, y, z file) that has not been converted to a vector point table using the Create Points command in MapInfo Professional, do the following:
   - From the X-Column list, choose the column containing the x-coordinates for each point.
   - From the Y-Column list, choose the column containing the y-coordinates for each point.
   - Click the Projection button and choose the coordinates system of the location data.
8. In the Enter Data Description box, type an annotation (maximum 31 characters). This will be carried as a header in the grid file.
9. To set the unit of measurement for the z-value, do one of the following:
   - From the Unit Type list, choose the appropriate unit of measurement of the z-value.
   - In the Enter User Defined Type box, type a user-defined unit of measurement.
10. If you want to include only non-zero records, select the **Ignore Records Containing Zero** check box.
11. Click **Next**. A dialog box specific to the type of interpolation opens.

Each dialog box is discussed in the following sections.

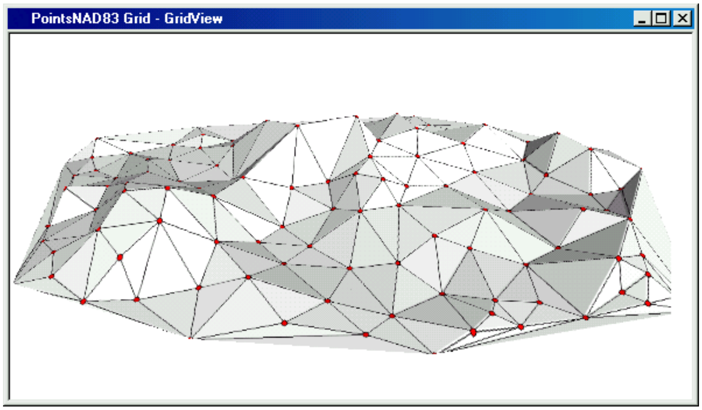
Triangulation with Smoothing

Triangulation is a process of grid generation that is usually applied to data that requires no regional averaging, such as elevation readings. The surface created by triangulation passes through (honours) all of the original data points while generating some degree of “overshoot” above local high values and “undershoot” below local low values. Elevation is an example of point values that are best “surfaced” with a technique that predicts some degree of over- and under- estimation. In modeling a topographic surface from scattered elevation readings, it is not reasonable to assume that data points were collected at the absolute top or bottom of each local rise or depression in the land surface.



Using the triangulation technique, the interpolated surface passes through the original data points. However, peaks and valleys will extend beyond the local maximum and minimum values.

Triangulation involves a process whereby all the original data points are connected in space by a network of triangular faces, drawn as equilaterally as possible. This network of triangular faces is referred to as a Triangular Irregular Network (TIN) as shown in the next figure. Points are connected based on the nearest neighbour relationship (the Delaunay criterion) which states that a circumcircle drawn around any triangle will not enclose the vertices of any other triangle.



A three dimensional view of a Triangular Irregular Network (TIN).

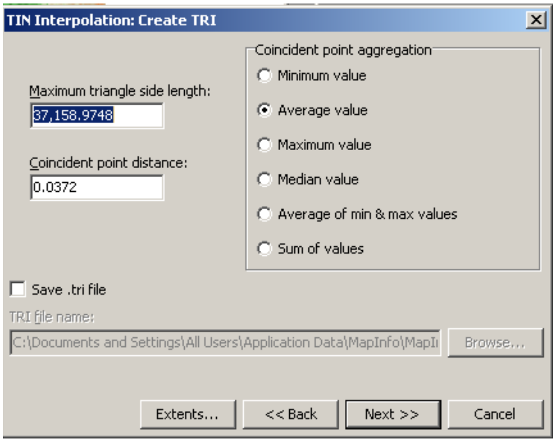
A smooth grid surface is then fitted to the TIN using a bivariate fifth-order polynomial expression in the x- and y- direction for each triangle face. This method guarantees continuity and smoothness of the surface along the sides of each triangle and smoothness of the surface within each triangle. The slope blending algorithm is designed to calculate new slope values for each of the triangle vertices (such as each point of the data) where the influence of adjacent slopes in the blending calculation is weighted according to specified triangle properties.

Five properties of data point geometry and value greatly influence the ability of the slope blending algorithm to control smoothing of the TIN surface. These include the triangle centroid location, the triangle aspect ratio, the triangle area, the angle versus slope of the triangle, and the statistically-derived slope of a triangle vertex. For example, triangles with centroids farther from the vertex being solved have less influence on the slope calculation than triangles whose centroids are closer; similarly, triangles with greater areas have greater influence in the slope calculation than triangles with a smaller area. The end result is a smoothing process that significantly reduces the frequency of angular artifacts, representing remnants of the original TIN facets in the final gridded surface.

Exploring TIN Interpolation: Create TRI Dialog Box

Once the data points have been analyzed by Vertical Mapper and exported to a temporary file, the TIN Interpolation: Create TRI dialog box opens. Two features of the triangulation algorithm, designed to build the Triangular Irregular Network (TIN) surface, are particularly critical and should be carefully considered:

- Maximum Triangle Side Length
- Coincident Point Distance



The **Maximum Triangle Side Length** box enables you to limit the creation of triangles between points you consider too far apart. Depending upon the geographic distribution of points in a file, this distance setting will restrict the generation of unnecessarily large and/or sliver triangles and will limit interpolation between points across distances that may be inappropriate. For example, a set of field observations may contain large gaps in the coverage due to sampling problems. If the gaps are quite large, it would be inappropriate in most cases to interpret new values lying within these areas. Generally, you are expected to have an understanding of the distribution characteristics of your data in order to enter an appropriate value. The default value is usually appropriate for most data sets.

Note Use the Ruler in MapInfo Professional to examine your point data file and measure gaps in the distribution of points over the map area. Use this information as a guideline in entering an appropriate value for the maximum triangle side length.

The **Coincident Point Distance** box enables you to group or aggregate data points into a single new point with a recalculated value. As the distance setting becomes greater, the number of points aggregated correspondingly increases. This may be appropriate in dealing with highly variable and irregularly distributed data. By increasing the coincident point distance, you prevent the creation of very small or sliver triangles that result in artifacts in the output grid.

The **Coincident Point Aggregation** section enables you to define the mathematical expression for handling aggregated data. For example, choosing a large coincident point distance and clicking the **Average Value** option button will create a new set of data points for triangulation, spaced approximately according to the distance setting and with recalculated values based on the average of all points in each coincident point area. New points are placed at the geometric centre of the original group.

The **Save TRI File** check box enables you to save the TIN surface for future gridding.

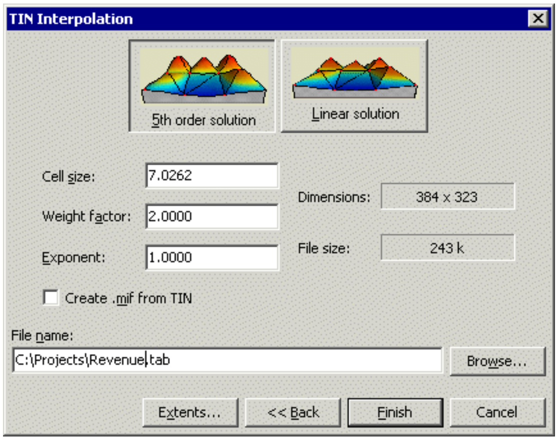
The **TRI File name** box enables you to enter a new file name for the .tri file.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Exploring the TIN Interpolation Dialog Box

The grid creation process consists of overlying a regularly-spaced network of equal-sized cells on top of the triangulated (TIN) surface and then estimating a new value for each cell based on a combination of variables. The variables include original point values, slope of individual and adjacent TIN faces, and distance to neighbouring triangles. The end result is a smoothed surface that passes through all of the original data points.

The TIN Interpolation dialog box enables you to set a number of user-defined parameters in the algorithm that control the grid generation from the TIN surface.



The **5th Order Solution** button enables you to apply a complex polynomial expression to the calculation of each grid node value. The calculation is based on solving a number of slope derivatives. The result is a more highly smoothed surface that displays minor angular artifacts from the original TIN.

The **Linear Solution** button enables you to calculate the grid values directly from the TIN surface; therefore, no derivative slope solution is applied. The result is a grid surface that exactly duplicates the angular appearance of the TIN.

The **Cell Size** box enables you to set the cell size in real units. The value chosen should be a compromise between the degree of resolution required for analysis and visualization purposes and the processing time and file size. The default value is calculated by dividing the diagonal extent of the point file by 500, which is considered an optimum number based on the average computing power available.

The **Weight Factor** box enables you to define the degree of influence imposed by neighbouring triangles in calculating a slope value for each triangulation point. Increasing the weighting factor will increase the relative influence of more distant triangle faces and will result in a greater degree of smoothing of the derived surface. Values range between one and 100.

The **Exponent** box enables you to set the variable that defines the exponential decay of influence of neighbouring triangle faces the farther they lie from each point. Increasing the exponent will decrease the relative influence of more distant triangles. Values range between one and 20.

Note The default values for the weight factor and the exponent are appropriate for most data. However, you should experiment with various settings in order to evaluate which variable generates the most appropriate representation of your data.

The **Create MIF from TIN** check box enables you to save the .tri file as a MapInfo format region file (.mif). You can then view the triangulated surface in a Map window as a wireframe representation using the Import command from the Table menu in MapInfo Professional.

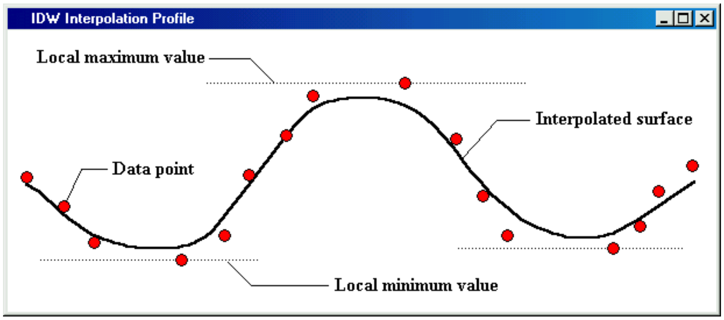
The **File name** box enables you to enter a new file name for the .mif file.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Inverse Distance Weighting Interpolation

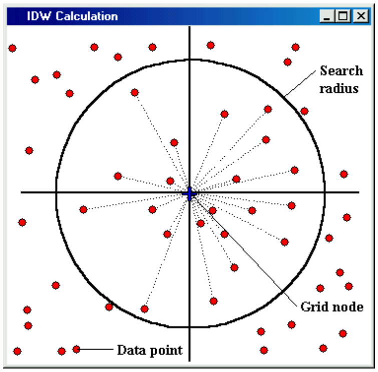
Inverse Distance Weighting (IDW) interpolation is a moving average interpolation technique that is usually applied to highly variable data. For certain data types, it is possible to return to the collection site and record a new value that is statistically different from the original reading but within the general trend for the area.

Examples of this type of data include environmental monitoring data such as soil chemistry and consumer behaviour observations. It is not desirable to honour local high/low values but rather to look at a moving average of nearby data points and estimate the local trends.



The interpolated surface, estimated by using a moving average technique, is less than the local maximum value and greater than the local minimum value.

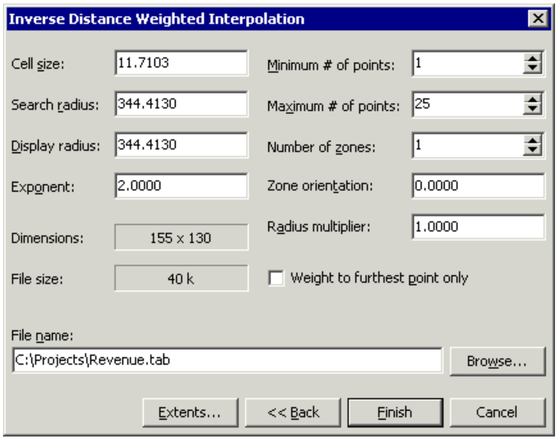
The IDW technique calculates a value for each grid node by examining surrounding data points that lie within a user-defined search radius. Some or all of the data points can be used in the interpolation process. The node value is calculated by averaging the weighted sum of all the points. Data points that lie progressively farther from the node influence the computed value far less than those lying closer to the node.



A radius is generated around each grid node from which data points are selected for use in the calculation.

Exploring the Inverse Distance Weighted Interpolation Dialog Box

The Inverse Distance Weighted Interpolation dialog box enables you to set a number of user-defined parameters in the algorithm.

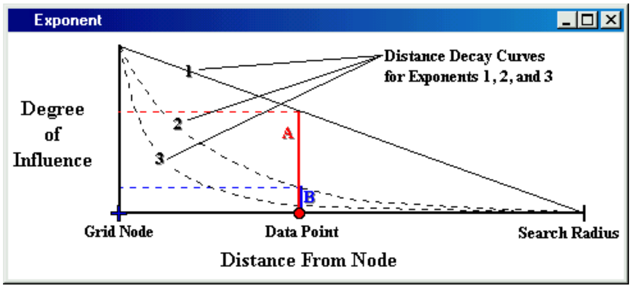


The **Cell Size** box enables you to set the cell size in real units. The grid dimensions (in cell units) vary inversely with cell size: the smaller the cell, the larger the grid file. The value you choose should be a compromise between the degree of resolution required for analysis and visualization purposes and the processing time and file size. The default value is calculated by dividing the diagonal extent of the point file by 300, which is considered an optimum number based on the computing power required to solve this slightly more complex algorithm.

The **Search Radius** box enables you to define the maximum size, in map units, of a circular zone centred on each grid node within which point values from the original data are averaged and weighted according to their distance from the node. You can also define the minimum and maximum number of data points averaged within each zone. The default setting is calculated as four times average point density.

The **Display Radius** box enables you to define the size, in map units, of a circular zone centred on each grid node within which an original data point must lie in order for that grid cell to be assigned a value and be displayed. This setting is used to control the extent of grid creation in areas of the map that do not contain data, such as the outer margins of the map area. The default setting that appears is the same as the search radius, which is appropriate for most data. However, if the original data points are characterized by very patchy distribution with occasional large gaps containing no sample sites, a smaller display radius should be considered in order to avoid interpolation of grid values within areas that do not contain data.

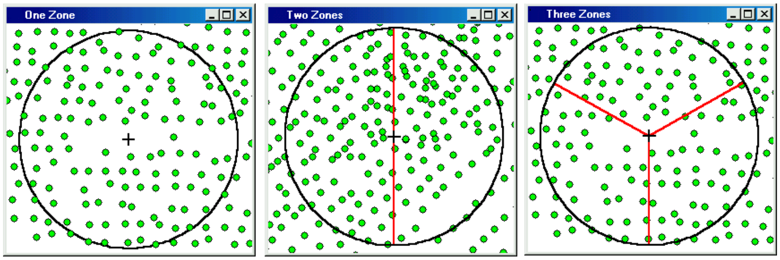
The **Exponent** box enables you to define the exponential rate of decay of influence by neighbouring points that lie farther from the grid node. Increasing the exponent will decrease the relative influence of more distant neighbours. Values range between one and 10 with a value of two being appropriate for most data.



As the exponent increases, the influence a data point has on the calculated grid node decreases. In this example, the data point has the amount of influence indicated by A; however, if the exponent is increased to two, the amount of influence B is dramatically reduced.

The **Minimum # of Points** box enables you to define the minimum number of points per zone, and the **Maximum # of Points** box enables you to define the maximum number of points per zone. This refers to the minimum and maximum number of data points searched for and averaged to calculate a single grid node value.

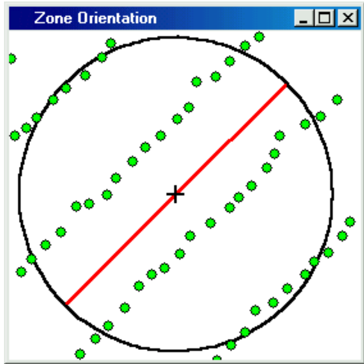
The three options that follow (Number of Zones, Zone Orientation, and Radius Multiplier) define the manner in which data points are averaged for each estimated calculation. The **Number of Zones** box enables you to define the number of equally sized partitions by which each search area may be divided.



The search area divided into one, two, and three equal zones.

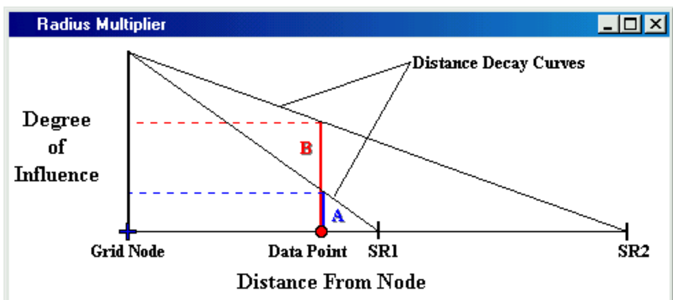
This may be useful if the data contains points that are clustered along a particular orientation such as data collected along widely separated survey lines. An equal number of points are averaged from each half, quadrant, or octant of the search area. In the majority of point files, the most appropriate number of zones is one. You should choose more than one zone only if you are confident that the data is clustered in a linear orientation.

The **Zone Orientation** box enables you to define unique orientation parameters in the creation of search area zones, which is appropriate only if data points are preferentially clustered in linear zones. This value is expressed in azimuth degrees.



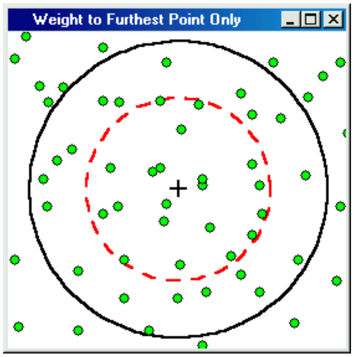
The search area is divided into two zones and a zone orientation set to match the distribution of the data points. This forces Vertical Mapper to use data points from two different sample paths when estimating grid values that lie between paths. The minimum and maximum number of points settings apply for each zone. In this figure, the zone orientation is set to 45 degrees.

The **Radius Multiplier** box enables you to define a weighting factor that determines the degree of influence of neighbouring points that lie successively farther from the grid node. A radius multiplier of one means a point located at the edge of the search radius has a weight of zero in the averaging calculation. A radius multiplier of two has the effect of doubling the distance used for weighting the points. Increasing the radius multiplier value is a way of increasing each point's influence on the calculation of a grid node without increasing the size of the search radius.



When the radius multiplier is set to one and the search radius at SR1, the data point will have the amount of influence, represented by A, on the grid node value. If the radius multiplier is set to two (two times the search radius distance, SR2), the data point will have an increased amount of influence represented by B. Only points within SR1 will be selected and used in the grid node calculation; however, the amount of influence they have can be controlled by the radius multiplier.

The **Weight to Furthest Point Only** check box enables you to apply a decay curve (exponent) based on the distance to the farthest point used in the calculation. This point is defined by the **Maximum # of Points** setting.



With the search radius defined by the Search Radius setting, the Weight to Furthest Point Only check box enabled, and the maximum number of points set to 20, the decay curve will be based on the search radius indicated by the dotted line.

Note .The default values for the search criteria and Number of zones, Zone orientation, and Radius Multiplier options are appropriate for most data.

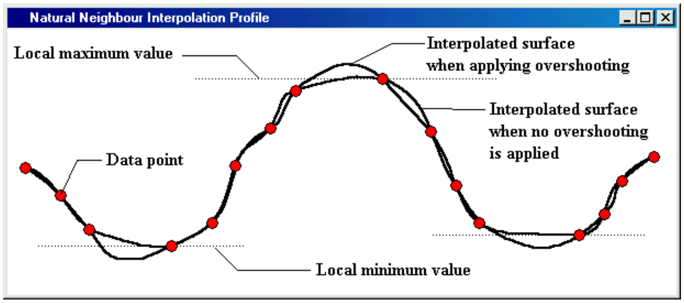
The ***File name*** box enables you to enter a new file name.

The ***Extents*** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Natural Neighbour Interpolation

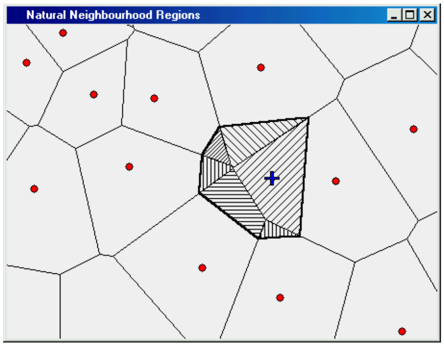
Natural neighbour interpolation is a geometric estimation technique that uses natural neighbour regions generated around each point in the data. This technique is particularly effective for dealing with a variety of spatial data themes exhibiting clustered or highly linear distributions.

The natural neighbour technique is designed to honour local minimum and maximum values in the point file and can be set to limit overshoots of local high values and undershoots of local low values. This technique thereby enables the creation of accurate surface models from data that is very sparsely distributed or very linear in spatial distribution.



The interpolated surface is tightly controlled by the original data points by honouring the value at each point. It also provides the option to over- or under- shoot point values.

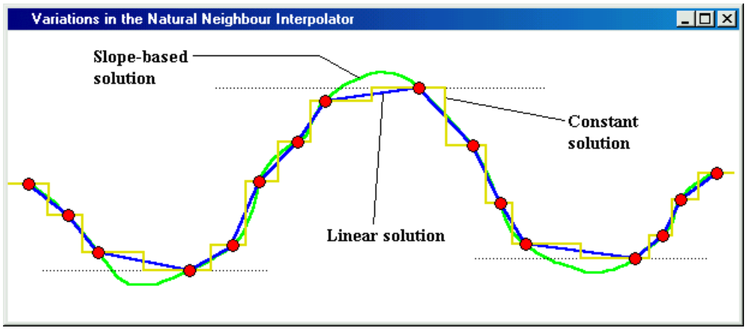
Put simply, natural neighbour interpolation makes use of an area-weighting technique to determine a new value for every grid node. As shown in the next figure, a natural neighbour region is first generated for each data point. Then, a new natural neighbour region is generated at every node in the new grid which effectively overlies various portions of the surrounding natural neighbour regions defining each point. The new grid value is calculated as the average of the surrounding point values proportionally weighted according to the intersecting area of each point.



A display of the natural neighbour regions around the point file as well as those created around a grid node.

You can choose one of three variations on the natural neighbour technique. The next figure illustrates the behaviour of each variation.

- A Constant Value Solution in which each grid node takes on the value of the underlying natural neighbour region.
- A Linear Solution where the grid value is determined by averaging the point values associated with surrounding natural neighbour regions and weighted according to the area that is encompassed by a temporary natural neighbour region generated around the grid cell (see the next figure).
- A Slope-Based Solution where the grid value is determined by averaging the extrapolated slope of each surrounding natural neighbour region and area weighted as in the linear solution. By examining the adjacent points, a determination is made as to whether that point represents a local maximum or minimum value. If it does, a slope value of zero is assigned to that value and the surface will honour that point by neither overshooting nor undershooting it.

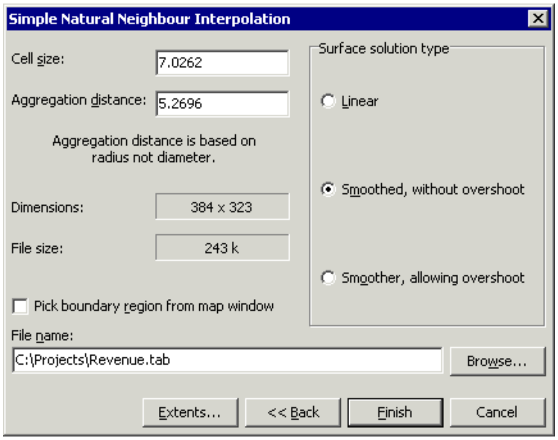


A graph showing the three variations of NN interpolation.

The Interpolation Wizard provides two methods of performing natural neighbour interpolation. The simple option offers the first-time user a two-step process for implementing the interpolation method. Many of the controls have been pre-set to generate the most appropriate surface given the distribution of points. The Advanced option enables you to fine-tune the grid surface.

Exploring the Simple Natural Neighbour Interpolation Dialog Box

The Simple Natural Neighbour Interpolation dialog box contains basic settings for building a grid file.



The **Cell Size** box enables you to set the cell size in real units. The grid dimensions (in cell units) vary inversely with cell size: the smaller the cell, the larger the grid file. The value chosen should be a compromise between the degree of resolution required for analysis and visualization purposes and the processing time and file size. The default value is calculated by dividing the diagonal width of the point file by 500, considered an optimum number based on the computing power required to solve this algorithm.

The **Aggregation Distance** box enables you to define the minimum separation allowed between data points before aggregation of the points is initiated. The aggregation technique makes use of the Forward Stepping method (see Forward Stepping Aggregation on page 178) where the aggregation distance is the radius of circular search zones centred on successive points in the data. Data points that fall within this search zone are mathematically averaged, and a new data point is placed at the geometric centre of the aggregated points. Aggregation is a useful method for quickly grouping points that may be virtually coincident. It is also used as a preliminary data smoothing technique where highly variable data points that are spaced closely together can be aggregated and new values calculated using averaging. If no aggregation is desired, set this value to zero. By default, the aggregation distance is set at 10 percent of the mean distance between the data points.

The **Surface Solution Type** section contains three options that are used to control the degree of smoothness applied to the new interpolated grid and refer to the different variations of the natural neighbour algorithm available.

- The **Linear** option calculates the value of a grid node using the average value of the surrounding points. The calculation is area-weighted to account for the relative influence of the surrounding points. The Linear Solution produces a profile that is slightly smoother than the constant value solution available with natural neighbour interpolation.

- The **Smoothed, without overshoot** option uses the slope-based solution discussed earlier. This variation of the Natural Neighbour method makes use of the slope of all adjacent points to calculate the area-weighted value of each grid node. Undershoot and overshoot are controlled by assigning a slope value of zero (horizontal) to all local minimum and maximum point values.
- The **Smoother, allowing overshoot** option uses the same slope-based method but allows overshooting and undershooting of local lows and highs, generating a much smoother surface.

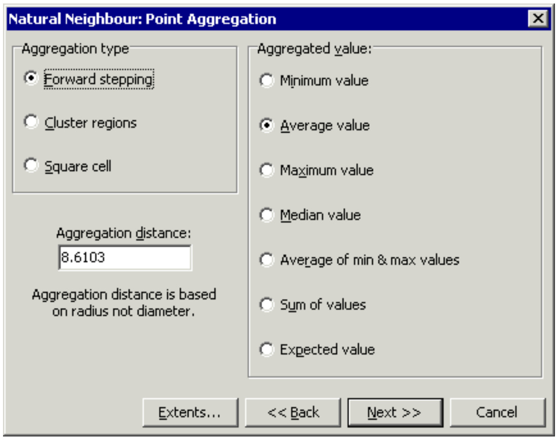
The **Pick Boundary Region from Map Window** check box enables you to choose a new boundary region as the outer hull of the point data.

The **File name** box enables you to enter a new file name.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Exploring the Natural Neighbour: Point Aggregation Dialog Box

When you choose the Advanced option in the Select Interpolation Method dialog box, you are given access to a variety of controls in the natural neighbour interpolation method that you can use to make subtle adjustments to the grid surface generated from a MapInfo points table.



The **Aggregation Type** section contains the three major aggregation techniques supported by Vertical Mapper. For more information about the operation and optimum usage of the forward stepping, cluster density, and square cell techniques, see **Techniques for Data Aggregation on page 172**.

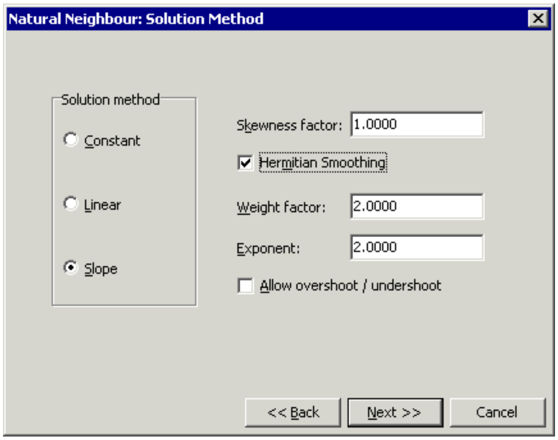
The **Aggregated Value** section enables you to choose how the new aggregated value is calculated. You can choose to average the aggregated values, average the minimum and maximum values, sum the values, or assign the minimum, the maximum, the median, or the expected value to the new aggregated point.

The **Aggregation Distance** box enables you to define the distance used to group points for aggregation. This distance has a different meaning for each of the three methods. For the cluster density and forward stepping methods, aggregation distance is defined by the radius of a user-defined circular search area centred on each aggregation cell. For the square cell method, aggregation distance is defined by the width of a square aggregation cell.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Exploring the Natural Neighbour: Solution Method Dialog Box

The final step in the Advanced Natural Neighbour technique involves choosing the solution method and setting a number of interpolation parameters.



The **Constant** option enables you to assign the exact value of the underlying natural neighbour region to the new grid node. The resulting grid is a very close approximation of the natural neighbour region map (Voronoi diagram) that can be generated from the original data points.

The **Linear** option enables you to assign the average z-value of each surrounding point weighted according to the area that is encompassed by a temporary natural neighbour region generated around the grid node.

The **Slope** option enables you to assign the area-weighted average slope of each surrounding point. By examining the adjacent points, a determination is made as to whether that point represents a local maximum or minimum value. If it does, a slope value of zero is assigned to that value, and the surface will honour that point by neither overshooting nor undershooting it.

The **Skewness Factor** box enables you to define a weighting factor used in the grid node calculation that can be used to assign greater or lesser influence to the data point that is closest to the grid node, such as the point whose natural neighbour area directly underlies the grid node. Allowable values range from 0.001 to 999. If the maximum allowable skewness factor is entered, the resulting grid will appear very similar to a grid generated using the constant value solution because the value at each grid node is almost entirely influenced by the point lying nearest to it.

The **Hermitian Smoothing** check box enables you to apply a weighting curve to the grid node calculation that effectively assigns greater influence to points that are closer to the grid node based on their proportionally larger intersecting areas. Conversely, Hermitian Smoothing will assign lower weighting to points that lie farther from the grid node based on their proportionally smaller intersecting areas.

Quantitatively, if the proportional natural neighbour area of a point is greater than 50 percent of the total grid node natural neighbour area, the Hermitian curve applies greater weight to this point. If the proportional natural neighbour area of a point is less than 50 percent of the total grid node natural neighbour area, the Hermitian curve applies lower weight to this point.

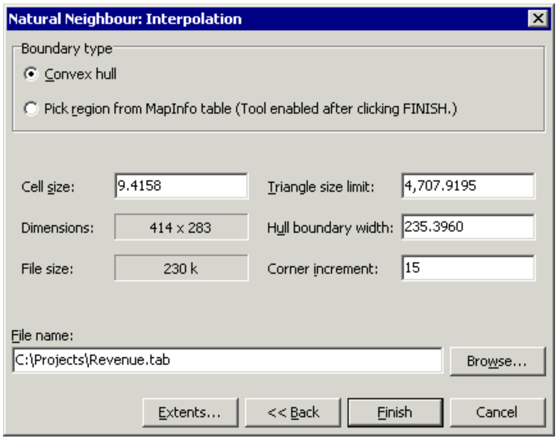
The **Weight Factor** box enables you to define the degree of influence imposed by neighbouring triangles in calculating a slope value for each data point. This is used only in the slope-based method of natural neighbour interpolation. Increasing the weighting factor will increase the relative influence of more distant data points.

The **Exponent** box enables you to set a variable, when calculating a slope value for each data point for the slope-based solution, that defines the exponential decay of influence by neighbouring points that lie farther from each point. Increasing the exponent will decrease the relative influence of more distant points.

The **Allow Overshoot/Undershoot** check box enables you to set the slope so that you can see values in the newly interpolated grid that exceed the range of the original point values.

Exploring the Natural Neighbour: Interpolation Dialog Box

After you have made the necessary adjustments to the grid surface in the Advanced Natural Neighbour Interpolation dialog box, you need to set a number of general parameters that control the grid building process.



The **Boundary Type** section refers to the selection of a hull that controls the outward extent of interpolation and forms the outermost boundary of the grid file. Two choices are available.

- The **Convex Hull** option enables you to set the natural boundary of the data defined by the outermost extent of the points. When you choose this option, the **Hull Boundary Width** and **Corner Increment** boxes become available.
- The **Pick Region from MapInfo Table** option enables you to choose a pre-defined MapInfo region as the point boundary. When you click the **Finish** button, you are prompted to pick a MapInfo table as a boundary region.

The **Cell Size** box enables you to set the cell size in real units. The grid dimensions (in cell units) vary inversely with cell size: the smaller the cell, the larger the grid file. The value chosen here should be a compromise between the degree of resolution required for analysis and visualization purposes and the processing time and file size. The default value is calculated by dividing the diagonal width of the point file by 500, considered an optimum number based on computing power required to solve this algorithm.

The **Triangle Size Limit** box enables you to specify a Delaunay triangulation setting to limit the creation of natural neighbour regions around adjacent points you consider too far apart. Depending upon the geographic distribution of points in a file, this setting will restrict the generation of unnecessarily large and/or extremely long, narrow regions. This will limit interpolation between points across distances that may be inappropriate. For example, a set of field observations may contain large gaps in the coverage due to sampling problems. If the gaps are quite large, it would be inappropriate in most cases to interpret new values lying within these areas. Generally, you are expected to have an understanding of the distribution characteristics of your data in order to enter an appropriate value.

The **Hull Boundary Width** box enables you to define a distance value in map units that is added to the convex hull of the data to define the outermost margin of the interpolated grid.

The **Corner Increment** box refers to the construction of the four corner segments of the hull of the data. The value entered represents the incremental arc angle of the segments that make up the imaginary polygon boundary at each corner. The smaller the value, the greater the number of segments that will be added and therefore the smoother the corner. The setting is measured in degrees and is limited to values between one and 30.

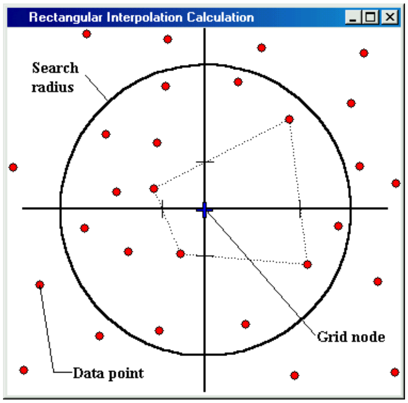
The **File name** box enables you to enter a new file name.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Rectangular Interpolation

Rectangular interpolation is usually applied to data that is regularly and closely spaced, such as points generated from another gridding application. This technique creates an interpolation surface that passes through all points without overshooting the maximum values or undershooting the minimum values.

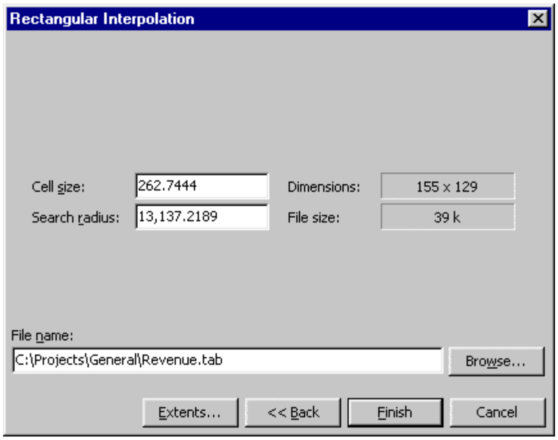
Rectangular interpolation locates the four nearest data points lying within a circular search zone, one from each quadrant, and connects them with a double linear rectangular framework (see next figure). An appropriate value is calculated for each node using the slopes of the connecting sides of the rectangle. However, in the absence of additional smoothing, linear artifacts are often generated across the surface when working with an irregular data point distribution.



A radius is generated around each grid node from which the closest data point in each quadrant is selected to be used in the calculation.

Exploring the Rectangular Interpolation Dialog Box

The Rectangular Interpolation dialog box enables you to specify parameters for the gridding algorithm. Rectangular interpolation uses a bilinear method for estimating individual grid node values. At each grid node, a circular search area of a set radius is generated and divided into quadrants. Four data points from the original point file are selected, one from each quadrant where each point is the nearest one to the node. A linear averaging solution is applied to the selected points to solve for the grid node value.



The **Cell Size** box enables you to set the cell size in real units. The grid dimensions (in cell units) vary inversely with cell size: the smaller the cell, the larger the grid file. The value you choose should be a compromise between the degree of resolution required for analysis and visualization purposes and the processing time and file size. The default value is calculated by dividing the diagonal extent of the point file by 200, which is considered an optimum number based on the computing power required to solve the rectangular algorithm.

The **Search Radius** box enables you to define the maximum size, in map units, of a circular zone centred on each grid node within which point values from the original data are selected based on the quadrant search technique. The default setting is calculated as a percentage of the total extent of the map area and is appropriate for most data.

The **File name** box enables you to enter a new file name.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Kriging Interpolation

Kriging is a geostatistical interpolation technique that considers both the distance and the degree of variation between known data points when estimating values in unknown areas. A kriged estimate is a weighted linear combination of the known sample values around the point to be estimated. Applied properly, kriging allows you to derive weights that result in optimal and unbiased estimates. It attempts to minimize the error variance and set the mean of the prediction errors to zero so that there are no overestimates or underestimates.

Included with the kriging function is the ability to construct a semivariogram of the data, which is used to weight nearby sample points. It also provides a means for you to understand and model the directional (for example, north-south, east-west) trends of your data. A unique feature of kriging is that it provides an estimation of the error at each interpolated cell, providing a measure of confidence in the modeled surface.

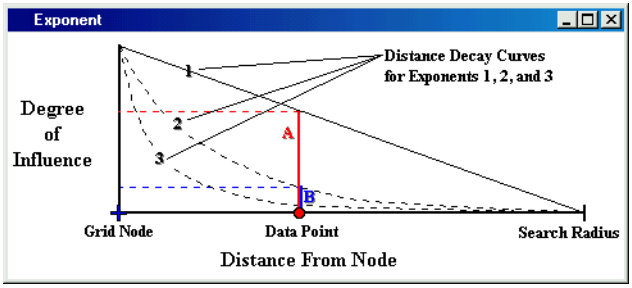
The effectiveness of kriging depends on the correct specification of several parameters that describe the semivariogram and the model of the drift (such as the mean value does or does not change over distance). Because kriging is a robust interpolation technique, even a naïve selection of parameters will provide an estimate comparable to many other grid estimation procedures. The trade-off for estimating the optimal solution for each point by kriging is computation time. Given the additional trial and error time necessary to select appropriate parameters, kriging should be applied where best estimates are required, data quality is good, and error estimates are essential.

Vertical Mapper provides three different methods of kriging interpolation: ordinary kriging, simple kriging, and universal kriging.

How Kriging Works

Kriging is a weighted moving average technique that is similar to Inverse Distance Weighting (IDW) interpolation. With IDW, each grid node is estimated using sample points that fall within a circular radius. The degree of influence each point has on the calculated value is based upon the weighted distance of each point from the grid node being estimated. In other words, points that are closer to the node will have a greater degree of influence on the calculated value than those points farther away.

The general relationship between the amount of influence a sample point has with respect to its distance is determined by the IDW Exponent setting, as shown below.

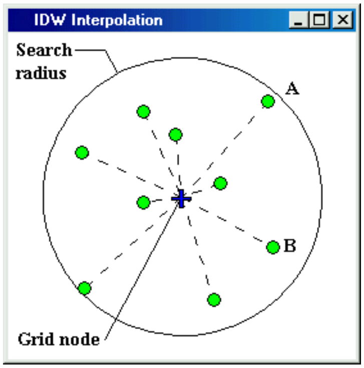


Decay Curves used by IDW interpolation.

The disadvantage of IDW interpolation is that it treats all points that fall within the search radius the same way.

For example, if an exponent of 1 is specified, a linear distance decay function is used to determine the weights for all points that lie within the search radius. This same function is used for all points regardless of their geographic orientation to the node (north, south, etc.) unless a sectored search is implemented.

Kriging, on the other hand, uses different weighting functions depending on the distance and orientation of sample points with respect to the node and the manner in which sample points are clustered.



This figure illustrates the influence points have, using IDW interpolation, on the calculated value based on the same distance decay function when one of the points is northeast of the grid node (point A) and the other point is southeast (point B). With kriging, the grid node may be calculated using a different weighting function for every point in the search radius.

Before interpolation begins, every possible distance weighting function is calculated by generating an experimental semivariogram and choosing a mathematical model to approximate its shape. The mathematical model provides a smooth, continuous function for determining appropriate weights for increasingly distant data points.

Understanding Kriging Techniques

Vertical Mapper provides three variations of kriging interpolation that you can apply in two forms, although they all operate in a similar way. The three methods are ordinary kriging, simple kriging, and universal kriging, and all three of these techniques can be applied in one of two forms: punctual or block.

Ordinary Kriging

This method assumes that the data has a stationary variance and a non-stationary mean value within the search radius. Ordinary kriging is highly reliable and is recommended for most data sets.

Simple Kriging

This method assumes that the data has a stationary variance and a stationary mean value and requires you to enter the mean value.

Universal Kriging

This method represents a true geostatistical approach to interpolating a trend surface of an area. The surface representing the drift of the data is built first, then the residuals for this surface are calculated. With universal kriging, you can set the polynomial expression used to represent the drift surface. The most general form of this expression is:

$F(x, y) = a_{20} * x^2 + a_{11} * xy + a_{02} * y^2 + a_{10} * x + a_{01} * y + a_{00},$

where a_{00} is always present but rarely set to zero in advance of the calculation. However, the other coefficients can also be set to zero. The recommended setting is a first degree polynomial which will avoid unpredictable behaviour at the outer margins of the data.

Punctual and Block Kriging

All three kriging interpolation techniques can be applied in one of two forms: punctual or block. The most commonly used is punctual kriging (the default), which estimates the value at a given point. Block kriging uses the estimate of the average expected value at a given location (such as a “block”) around a point. Block kriging provides better variance estimation and has the effect of smoothing interpolated results.

Using the Kriging Interpolation Technique

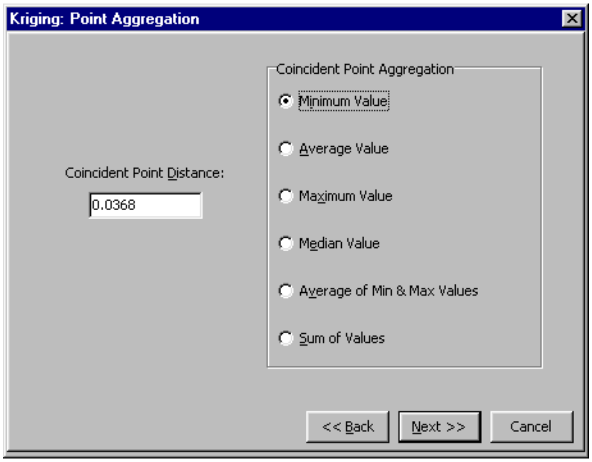
There are four basic steps in the kriging process:

- aggregate the data
- choose the kriging parameters
- complete the variogram analysis
- perform the kriging estimation

If you choose all the system defaults, the kriging type will be ordinary, the experimental semivariogram will be calculated, a model will be automatically fitted to the data, and kriging interpolation will be performed. However, experienced users will always spend some time fitting a model to the semivariogram. For more information about generating and interpreting a semivariogram, see **Semivariogram Analysis on page 154**.

Exploring the Kriging: Point Aggregation Dialog Box

The Point Aggregation dialog box contains settings that will process the point file and remove any coincident points. Coincident points must be removed in order to successfully interpolate a surface.

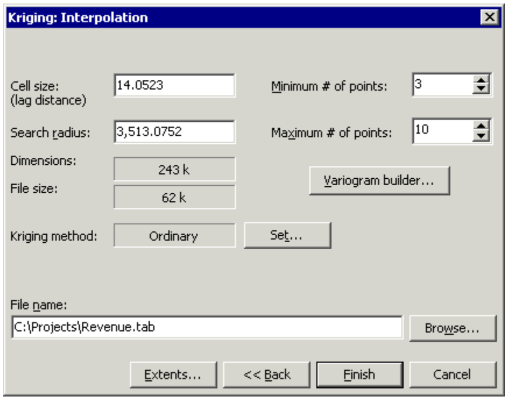


The **Coincident Point Distance** box enables you to group or aggregate data points into a single new point with a recalculated value. As the distance setting becomes greater, the number of points found within each circular area will correspondingly increase. This may be appropriate in dealing with highly variable and irregularly distributed data.

The **Coincident Point Aggregation** section enables you to define the mathematical expression for handling aggregated data. For example, if you choose a large coincident point distance, then choose the Average Value option, a new set of data points for interpolation is created, spaced approximately according to the distance setting and with recalculated values based on the average of all points in each coincident point area. New points are placed at the geometric centre of the original group.

Exploring the Kriging: Interpolation Dialog Box

The Kriging Interpolation dialog box enables you to set a number of user-defined parameters.



The **Cell Size** (lag distance) box enables you to set the cell size in real units. The grid dimensions (in cell units) vary inversely with cell size: the smaller the cell, the larger the grid file. The value chosen here should be a compromise between the degree of resolution required for analysis and visualization purposes and the processing time and file size. The default value is calculated by dividing the diagonal width of the aggregated point file by 250, considered an optimum number based on the computing power required to solve this computationally intensive algorithm.

The **Search Radius** box enables you to define the maximum size, in map units, of a circular zone centred on each grid node within which point values from the original data will be used in the calculation. You can also define the minimum and maximum number of data points in the Search Criteria section. The default setting is calculated as the diagonal distance through the minimum bounding rectangle of the point data. This setting will automatically change based on the results of the semivariogram analysis. The analysis is performed when you click the Variogram Builder button.

The **Minimum # of Points** and **Maximum # of Points** lists enable you to set the minimum number of points that must be found inside the search radius in order to assign a calculated value to a grid node, and the maximum number of points that will be used in the calculation. The default values of three and 10 respectively are appropriate for most data. You must keep in mind that if the maximum number of points in this setting is doubled, the processing time will increase by a factor of eight.

The **Set** button enables you to choose one of three different types of kriging: ordinary kriging, simple kriging, and universal kriging. The default method is set to ordinary kriging which is suitable for most data. Regardless of the kriging type employed, block kriging can also be implemented.

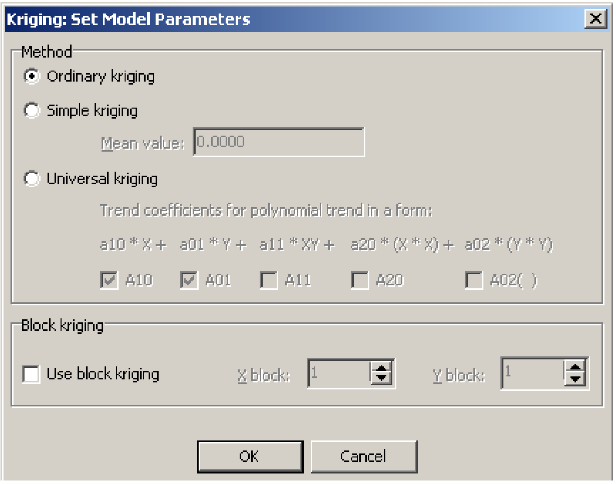
The **Variogram Builder** button enables you to build a semivariogram of the data, enabling you to match or tune a mathematical model to the experimental semivariogram. For more information, see **Semivariogram Analysis on page 154** and **Exploring the Variogram Dialog Box on page 159**.

The **File name** box enables you to enter the file path and name of the new grid that will be created. With the kriging interpolation technique, two grids are created. The first is the interpolated surface grid and the second is a grid of the estimated variance at each grid cell. The variance grid will have the same file name as the surface grid but will have “_var” appended to the end of the name. Both grids are placed in the same folder.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Exploring the Kriging: Set Model Parameters Dialog Box

When you click the Set button in the Kriging Interpolation dialog box, the Set Model Parameters dialog box opens. This dialog box provides additional settings for each kriging method.



The following options are available in the **Method** section:

- **Ordinary Kriging** enables you to generate an interpolated surface with the default settings.
- **Simple Kriging** enables you to specify the mean value used in the interpolation calculation.
- **Universal Kriging** enables you to modify the polynomial expression used to approximate the drift in the data. A detailed explanation of the concept of a regionalized variable and drift versus residual values is beyond the scope of this manual. For more information, refer to the suggested readings at the end of this chapter.

The following option is available in the **Block Kriging** section:

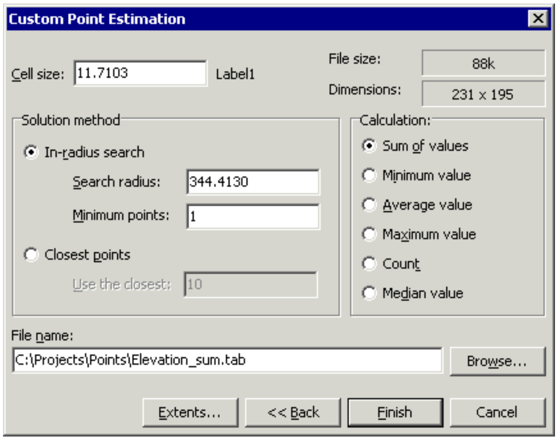
- The **Use Block Kriging** check box enables you to specify the x-block and y-block settings that define the level of discretization for the area around every point. Clear the check box to apply punctual kriging.

Custom Point Estimation

The Custom Point Estimation technique is similar to the IDW (Inverse Distance Weighting) technique, in which grid values are calculated based upon the points found within a predefined search radius. The main difference between these two techniques is that using the Custom Point Estimation technique, you can choose from six different calculations to perform on data points.

Exploring the Custom Point Estimation Dialog Box

The Custom Point Estimation dialog box enables you to specify parameters that are specific to the point estimation calculation.



The **Cell Size** box enables you to specify the cell size in the coordinate unit of the point file to be processed.

The **Search Radius** box enables you to specify the size of a radius land area in which points are to be calculated for.

The **Solution Method** section contains the following options.

- The **In-Radius Search** option enables you to define the search radius and the minimum number of points within the search radius. If the number of points found is less than the minimum value as defined, a null value is returned.
- The **Closest Points** option enables you to define the number of closest points that are included in the calculation.

The **Calculation** section contains the following options:

- The **Sum of Values** option returns the sum of the values of all the points within the search area or within the number of selected points.
- The **Minimum Value** option returns the lowest value of all the points within the search area or within the number of selected points.
- The **Average Value** option returns the average value calculated by averaging all valid points found within the search area or within the number of selected points.
- The **Maximum Value** option returns the highest value of all the points within the search area or within the number of selected points.

- The **Count** option counts the number of points found within the search area or within the number of selected points.
- The **Median Value** option returns the middle value of all the points within the search area or within the number of selected points.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Suggested Reading on Interpolation Techniques

For further information that covers interpolation and contouring techniques, including an exhaustive reference list, refer to:

Watson, D.F., 1992: *Contouring: A Guide to the Analysis and Display of Spatial Data*. Elsevier Science Inc., Tarrytown, New York, NY.

For information on the fifth-order bivariate interpolation applied to the TIN solution, refer to:

Akima, H., 1978: A Method for Bivariate Interpolation and Smooth Surface Fitting for Irregularly Distributed Data Points. *ACM Transactions on Mathematical Software*, v.4, no. 2, pp. 148-159.

Natural Neighbours

Gold, C.M. and Roos, T., 1994: *Surface Modelling with Guaranteed Consistency – An Object-Based Approach in Nievergelt*, J., Roos, T., Schek, H.-J., and Widmayer, P. (eds.), IGIS' 94, Proceedings of the International Workshop on Advanced Research in Geographic Information Systems, Lecture Notes in Computer Science 884; Springer-Verlag, pp. 70-87.

Kriging

Olea, R., editor, 1991: *Geostatistical Glossary and Multilingual Dictionary*. Oxford University Press, New York, NY.

Deutsch, Clayton V., and Journel, Andre G., 1992: GSLIB, *Geostatistical Software Library and User's Guide*. Oxford University Press, New York, NY.

Isaaks, Edward H., and Srivastava, R.Monah, 1989: *Applied Geostatistics*. Oxford University Press, New York, Oxford.

Goldberger, A., 1962: *Best Linear Unbiased Prediction in the Generalized Linear Regression Model*. JASA, 57: pp. 369-375.

4

Creating Grids Using Spatial Models

This chapter describes all the basic commands associated with the creation of numeric grid files using modeling techniques.

This chapter covers how to:

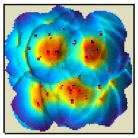
- create a Location Profiler model
- create a Trade Area Analysis model using a single site or multiple sites

In this chapter:

Creating Grids using Spatial Models

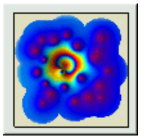
Modeling techniques create grids of derived values. The modeling wizard enables you to create grid files using two basic techniques.

Location Profiler



The Location Profiler creates a model where cell values are calculated as the average distance to all points found within a user-defined radius.

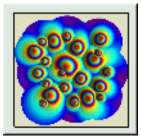
Trade Area Analysis - Single Site



Trade area analysis enables you to generate trade areas around stores or other services based on the probability of an individual patronizing a particular location.

The trade area for a single site is calculated using the distance customers live from the site and the ability of the site to attract them. Cell values represent the probability of a customer patronizing the site.

Trade Area Analysis - Multiple Site



Trade areas for multiple sites are calculated using the distance customers live from the site and the ability of the site to attract them. Cell values represent the probability of a customer patronizing any one of the sites in the selection.

Understanding the Location Profiler Model

The Location Profiler computes and averages the distance to a series of points from anywhere within a map area. The algorithm generates a grid, where at each cell a value is calculated that represents the average distance to all point locations surrounding that cell and lying within a defined search radius. You are creating a geographic profile of an area that measures proximity to a series of sites using spatial relationships (spacing, distribution, and density). By identifying those locations on the map that are the shortest average distance to all or some of the sites, the grid file can be used to represent geographic centres of activity. The model can be further refined to take into account weighting factors that specify the relative influence of each site compared with those surrounding it.

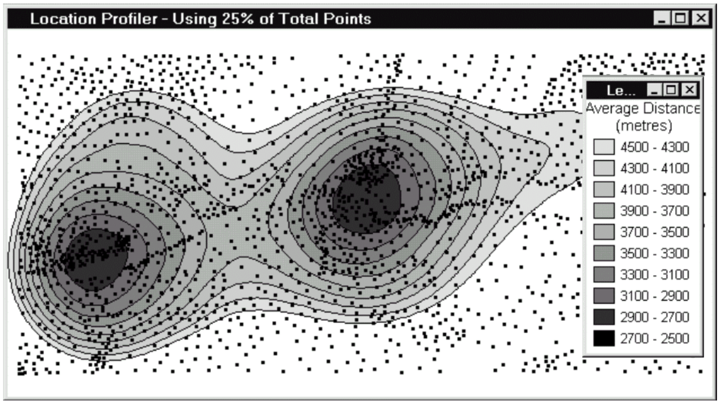
Three settings are critical to making the Location Profiler a truly effective modeling tool:

- the number of points to which the distance is computed from each grid node
- the relative influence or weighting of each point in the averaging calculation
- the distance decay function of the weighting factor

Setting the Number of Points

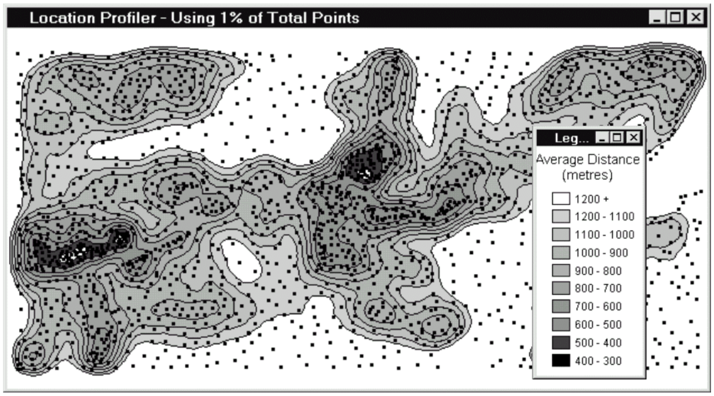
At each cell of the grid, the Location Profiler measures the distance to every point lying within the search area and calculates the average value. However, the pattern of geographic centres generated by the process will vary depending upon the number of points used for each grid cell calculation. The number of points is determined by defining a search area around each grid cell to select points, and defining the minimum and maximum number of points to be used for each calculation.

The next two figures show the result of varying the number of points used by the Location Profiler for each grid cell calculation. By default, the algorithm sets the search radius and the maximum number of data points to include 100 percent of the points in the table. Using the default settings thereby creates a grid that defines the most central region of the point location database.



This illustration shows the geographic profile of a table of point locations (represented by black dots) calculated using 25 percent of the total number of points. Sites lying within the more central contour regions are closer to the surrounding points than sites lying within the outermost regions.

Reducing the number of points involved in each grid cell calculation, by decreasing both the search radius and the maximum number of data points, tends to create a more complex profile that highlights local zones of greater point density within the overall distribution of points. This may be useful if, for example, you want to highlight local areas of greater site density across a geographic area.



This illustration shows the geographic profile of the same table of locations calculated using only one percent of the total points. The highlighted local concentrations of points give an effective visual representation of variations in point density across the map area.

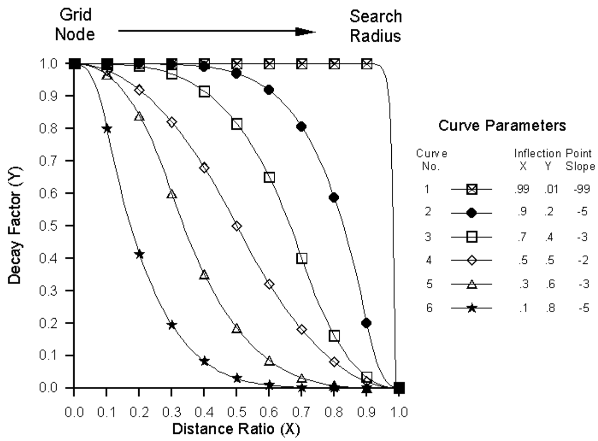
In addition to defining a search area that controls the number of sites used in the distance averaging calculation, you can also define an exclusion radius that is used to create a circular area immediately surrounding each grid cell within which site locations are excluded from the calculation.

Using Point Weighting

The Location Profiler allows the use of weighting values attached to points. If all data points are considered to be of equal importance, then their weighting values will all be equal (typically one). If some data points are considered to be more important, or perhaps more reliable, correspondingly higher values may be attributed to them. In real terms, the weighting factor may represent a relative confidence value assigned to each point if, for example, each site represented a human observation. The factor may also be a measure of frequency if, for example, each site represented a freight company's customer and showed the number of deliveries per month. In each case, the distance calculation is treated as a multiple of the weighting factor. For example, if the weight represented frequency of deliveries, then each distance calculation would be handled as a multiple measurement based upon the frequency value.

Using Distance Decay Functions

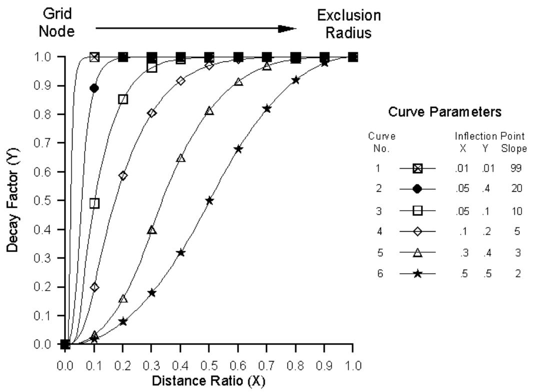
A distance decay function can be applied to all points in the location table assuming that a weighting factor has also been assigned. Distance decay can be used in the Location Profiler model if it is assumed that the data points close to the grid point are more relevant than those lying farther from the grid point. In other words, you must decide if it is reasonable to expect that, as the distance to the data points approaches the outer edge of the search radius, the contribution of these values approaches zero. A decay value of one is suggested for data points close to the grid node, and it falls to zero for data points closer to the edge of the search radius. A distribution of this type typically has an inflection point somewhere along its length. An inflection point is the point along the distribution where the slope either stops increasing in value or stops decreasing in value. For typical distance decay curves, as shown in the next figure, the slope is zero where x = 0 and gradually decreases, becoming more negative, until the inflection point is reached. At this point, the slope gradually increases, becoming more positive and returning to zero again as x approaches one.



Shown here are six search radius distance decay curves supported by the Location Profiler. Values along the x-axis represent a ratio obtained by dividing the distance between the grid point and data point by the search radius. Curve 3 represents the most typical search radius decay function that can be applied to the widest variety of modeling parameters.

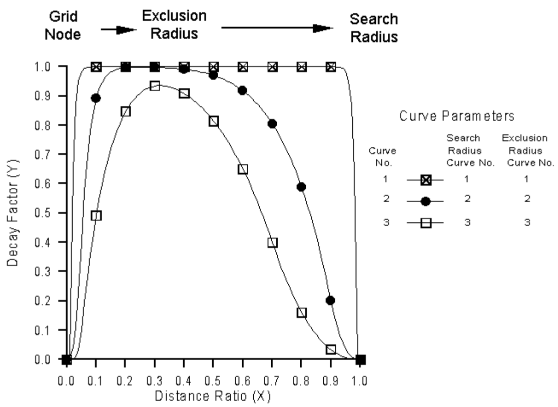
A similar decay function can be applied to the exclusion zone of the grid cell but using a curve with a positive slope. In this case, the slope is zero where x=0 and gradually increases, becoming more positive, until the inflection point is reached and then gradually decreases and returns to zero as x approaches one.

Various methods can be used to obtain the distributions shown in the following figures. Most techniques require that you specify the location (x- and y- coordinates) of the inflection point and the slope at this inflection point. Given an appropriate distance decay curve, a decay factor can therefore be determined for any distance value measured between the grid cell and a weighted point location. When x = 0 (at the grid cell assuming no exclusion radius setting), then y = 1 (no distance decay factor), and the slope is 0. When x=1 (at the outer edge of the search radius), then y = 0 (100 percent decay), and again the slope is 0.



Shown here are six exclusion radius distance decay curves supported by the Location Profiler. Values along the x-axis represent a ratio obtained by dividing the distance between the grid point and data point by the exclusion radius. Curve 3 represents the most typical exclusion radius decay function that can be applied to the widest variety of modeling parameters.

The y-axis indicates the decay factor associated with the data point. This decay factor, which is always between zero and one, is then multiplied by the weighting factor associated with the data point to obtain the weighted value. This weighted value is then multiplied by the measured distance between the grid point and data point, to obtain a weighted distance. The weighted average distance for the grid point is obtained by summing all the weighted distances for each data point, then divided by the sum of all the weighted values of the data points.



This figure shows the blending relationship between distance decay functions relating to the exclusion zone and the search zone. A continuous function can be created by choosing complementary curves that will distance weight all point locations lying between the grid node and the search radius.

Creating a Location Profiler Model

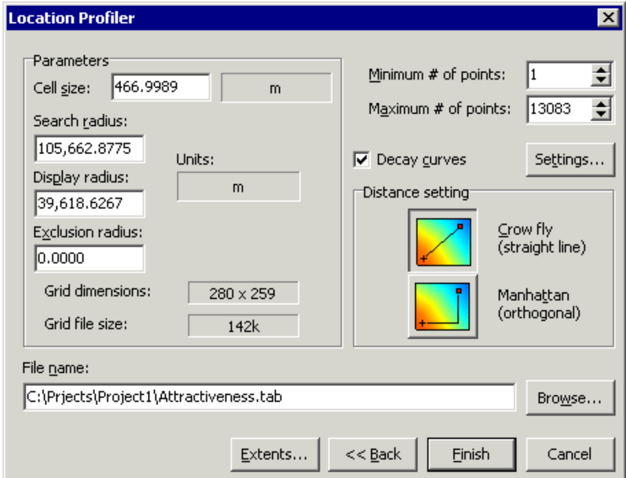
To create a Location Profiler Model:

1. From the Vertical Mapper menu, choose the **Create Grid > Modeling** command.
2. From the Select Modeling Method dialog box, choose the **Location Profiler** option.
3. Click **Next**.
4. In the Select Table and Column dialog box, click **Load Table** to add a table to the ***Table to Grid*** list if the table of points you want to use is not listed.
5. From the Select Table to Grid list, choose the appropriate MapInfo table of points that contains the geographic location of each point.
6. From the Weight Column list, choose the column that contains the weighting value for each site record. If no weighting value has been assigned to the point records, choose the **No Weight** entry from the list. Weighting factors are values that specify the relative influence of each site compared with those surrounding it.
7. To use an unmapped MapInfo data file (an x, y, z file) that has not been converted to a vector point table (using the Create Points command in MapInfo Professional), do the following:
   - From the X Column list, choose the column containing the x-coordinates for each point.
   - From the Y Column list, choose the column containing the y-coordinates for each point.
   - Click the **Projection** button and choose the coordinate system of the location data.
8. In the Data Description box, type an annotation (maximum 31 characters). This will be carried as a header in the grid file.
9. Click **Next**, enter the required information in the dialog box, then click the **Finish** button.

Once the grid is created, it appears in a Map window with a default colour palette applied. You can change the colour range assigned to the grid file. For more information, see **Using the Grid Colour Tool for Numeric Grids on page 97**.

Exploring the Location Profiler Dialog Box

Once the points and their appropriate weighting values have been exported to a temporary file, the Location Profiler dialog box opens. This dialog box summarizes information relating to the point table and enables you to set a number of user-defined parameters in the Location Profiler algorithm.



The **Cell Size** box enables you to define the cell size in the native coordinate system units of the map. The grid dimension (in cell units) varies inversely with cell size: the smaller the cell, the larger the grid file. The value you choose should be a compromise between the degree of resolution required for analysis, the processing time, and file size. The grid dimensions (in cell width and height) and the grid file size (in kilobytes) are displayed and updated as the cell size is adjusted.

The **Search Radius** box enables you to define the maximum size of the circular zone centred on each grid node that is used to select point locations. For each point location lying within this search zone, the distance between the grid cell (the centre of the zone) and the point is measured. By default, the search radius is set to select all points in the point table. If the point table is large, reduce the number of grid calculations by increasing the cell size. Otherwise, decrease the search radius.

The **Display Radius** box enables you to define the size of the circular zone centred on each grid node within which a point must lie in order for that grid cell to be assigned a value and be displayed. This setting is used to control the extent of grid creation in areas of the map that do not contain data, such as the outer margins of the map area.

The **Units** field displays the unit of horizontal distance measurement used in the model calculation (cell size, search radius, and display radius). If the point table is set to a Latitude/Longitude coordinate system, then the cell size unit is in degrees and you can select and set the units for **Search Radius, Display Radius**, and **Exclusion Radius** in kilometres, miles, metres, or feet from the Unit list. All three parameters are updated instantly to the selected unit.

However, when applied to the Location Profiler model, all distance calculations are automatically converted to arc lengths corrected for earth curvature (oblate spheroid model).

The **Minimum # of Points** box enables you to set a lower limit on the number of point locations that must be found from each cell before a calculation is made. The default value is one.

The **Maximum # of Points** box enables you to set an upper limit on the number of point locations measured from each grid cell. Used in combination with the search radius setting, the Location Profiler model will stop searching for point locations once either setting has been maximized. The default setting is the total number of points found in the table.

The **Decay Curves** check box enables you to control the application of decay functions to the weighting values attached to each point location. If you don't choose a weighting column in the ***Select Table and Column*** **dia**log box, the decay curves **Settings** button is disabled. If a weighting column is chosen in the dialog box, you can choose to disable the decay functions by clearing the **Decay Curves** check box.

The **Distance Setting** section enables you to define how distances are measured, in map units, between each grid cell and the point locations. One of two techniques can be set in the Location Profiler model.

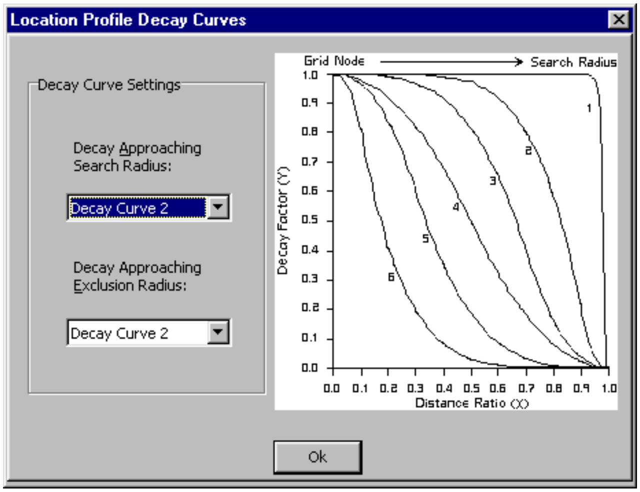
- **Crow Fly** (Straight Line) method measures the distance between the grid cell and the point using a straight line path.
- `Manhattan` (Orthogonal) method measures the distance between the grid cell and the point location using the x and y difference between the two sites, that is, it combines the total x-axis and y-axis distance that must be covered to reach a destination. For Latitude/Longitude data the distance is measured first travelling north-south then east-west. This option is provided to approximate distances when travelling on road networks that are oriented in a square grid pattern.

The **File name** box enables you to enter a new file name.

The **Extents** button displays a summary of the geographic size, the z-value range of the point database, and the data units.

Exploring the Location Profile Decay Curves Dialog Box

Distance decay should be applied only when it is assumed that data points lying close to the grid cell are more relevant than those lying farther away. In other words, as the distance from the grid node to the data points approaches the search radius, the relative contribution of these points to the distance averaging calculation should approach zero. A decay value of one (100 percent contribution) is defined for data points close to the grid node, falling to zero (zero percent contribution) for data points at a distance right at the search radius from the grid node. Decay functions can also be applied to the exclusion zone. Using the same model, the decay value is zero for points at the grid node (zero percent contribution) and increases to one for data points at a distance close to that of the exclusion radius (100 percent contribution).



The **Decay Approaching Search Radius** list enables you to choose one of the supported curves from the list for the calculation. Each decay function defines a smooth curve with a negative slope that always starts at coordinate (0, 1) with a slope of zero and always ends at coordinate (1, 0) with a slope of zero. The coordinates of the inflection point (where the slope of the curve either stops increasing in value or stops decreasing in value) and the slope at this point is recorded for each function.

- Decay Curve 1 defined by an inflection point at (.99, .01) having a slope of -99.
- Decay Curve 2 defined by an inflection point at (.9, .2) having a slope of -5.
- Decay Curve 3 defined by an inflection point at (.7, .4) having a slope of -3.
- Decay Curve 4 defined by an inflection point at (.5, .5) having a slope of -2.
- Decay Curve 5 defined by an inflection point at (.3, .6) having a slope of -3.
- Decay Curve 6 defined by an inflection point at (.1, .8) having a slope of -5.

The **Decay Approaching Exclusion Radius** list enables you to choose one of the supported curves from the list for the calculation. Each decay function defines a smooth curve with a positive slope that always starts at coordinate (1, 0) with a slope of zero and always ends at coordinate (1, 1) with a slope of zero. The coordinates of the inflection point and the slope of the curve at this point are recorded for each function.

- Decay Curve 1 defined by an inflection point at (.01, .01) having a slope of 99.
- Decay Curve 2 defined by an inflection point at (.05, .4) having a slope of 20.
- Decay Curve 3 defined by an inflection point at (.05, .1) having a slope of 10.
- Decay Curve 4 defined by an inflection point at (.1, .2) having a slope of 5.
- Decay Curve 5 defined by an inflection point at (.3, .4) having a slope of 3.
- Decay Curve 6 defined by an inflection point at (.5, .5) having a slope of 2

Note If none of the curves provided describes the decay characteristics of your data, create a curve by editing the vmxclude.cfg or the vmweight.cfg file located in the *All Users\Application Data\MapInfo\MapInfo\Vrtical Mapper\350* folder.

Understanding the Trade Area Analysis Model

The Trade Area Analysis models enable you to generate trade areas around stores or other services based on the probability of an individual patronizing a particular location. It is possible to identify market islands in a network of competing stores using criteria such as the attractiveness of a store and the distance one must travel to get to the store. Store attractiveness can be defined by parameters such as total floor space or shelf space, number of parking spaces, age of the store, or any combination of elements that defines its appeal. Using commercially available wealth and market profile information for potential customers located within each trade area, it is possible to estimate revenue for store locations and model the influence of competing stores or the effect of adding or removing stores.

Trade Areas

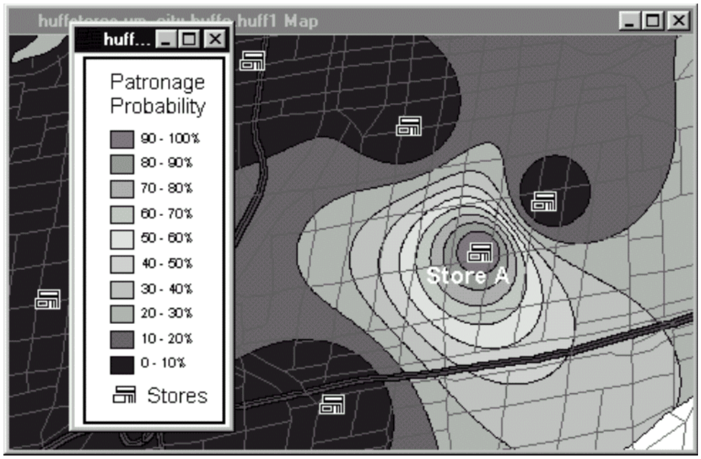
Trade areas can be defined by simple circular boundaries around a store location. These trade areas are easy to visualize and construct and provide a standard for comparing stores. Trade areas determined in this way do not, however, account for the existence of competing stores and assume that the store has a monopoly over customers within the area. There is the common sense concept that, all other things being equal, a person is likely to shop at a closer store rather than a more distant one. This concept of likelihood forms the basis for defining probabilistic trade areas.

Defining the likelihood or probability that a given customer will patronize a certain store requires the use of a spatial interaction model. Examining movement over space for activities such as shopping trips, commuting to work, or migration patterns, became popular with the use of gravity models in the 1960s and 1970s.

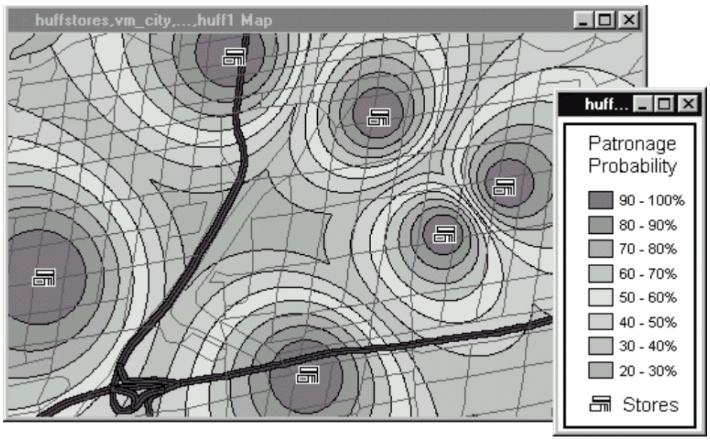
Gravity models are based on an analogy with Newton’s theory of gravitational attraction. The degree of attraction between two objects is based on the size of the objects and the distance between them. Objects that are closer together will exert a greater pull on each other compared with objects that are farther away. Larger objects have a greater gravitational pull than smaller objects. Refinements to the gravity model were made by Professor David Huff.

The Huff model is still one of the most popular models for predicting retail customer behaviour. It enables the mapping of the neighbourhoods from which each store derives its patronage. The answer is not a single circle or polygon but a probability surface or grid. This probability grid can be contoured to produce regions of patronage probability. The key feature implicit in the probability surface is that it accounts for overlapping trade areas.

Using a Huff model, you can calculate the probable trade area of a single store (next figure) or compute the patronage probability values for every store and extract the maximum probability value at each grid location. You can then use this information to determine areas where people are least serviced or areas where there is great competition for a customer’s business, that is, there is no preferred store location clearly winning a customer’s patronage.



This map displays the probability that a consumer will patronize Store A. The influence of all stores is taken into account in this model so that the locations of stores surrounding Store A influence the shape of the contours. Patronage probability decreases as the distance from the store increases.



This map displays the maximum probability at every potential customer location that a consumer will patronize any one store when presented with all stores.

Retailers can modify the Huff equation parameters by defining the attractiveness of each store competing for a customer's patronage and determining the rate at which the pulling power of a store drops off or decays with distance from the store. This produces a map estimating the probability that a potential customer (Customer) residing at a given grid location will patronize a particular store at a particular location (ReferenceStore) given a network of competing stores (Store1 to StoreN where N = the total number of stores including the Reference Store). This relationship is presented in the equation below.

$$\text{CUSTOMER PROBABILITY} = \frac{\dfrac{(\text{Attractiveness of } \textit{ReferenceStore})^{\alpha}}{(\text{Distance between } \textit{Customer} \text{ and } \textit{ReferenceStore})^{\beta}}}{\text{SUM of} \left[\dfrac{(\text{Attractiveness of } \textit{Store1})^{\alpha}}{(\text{Distance between } \textit{Customer} \text{ and } \textit{Store1})^{\beta}} \right] \text{for } \textit{Store1} \text{ to } \textit{StoreN}}$$

Where:

α = attractiveness coefficient estimated from empirical observations

β = distance decay coefficient estimated from empirical observations

This formula ensures that all the probabilities are between zero and one and sum to one when considering all the shopping destinations. The α and β coefficients allow you to account for non-linear behaviour when modeling the effects of store attractiveness or distance.

In terms of a supply and demand model, the resulting probability surface spatially represents the supply side of the equation. It does not directly take into account any underlying demand data such as consumer households or census tract information. To process the demand data so that revenue calculations can be made, it is necessary to update the probability regions (produced by contouring the probability grid) with any underlying income information you have.

Note Instead of updating the probability regions with overlying demographic (population/wealth) point data, update the demographic data table with values from the probability grids. Open the grid files and use the Point Inspection command from the Analysis menu in the Grid Manager to add the grid values to the demographic data table.

Creating a Trade Area Analysis Model

1. From the Vertical Mapper menu, choose the **Create Grid > Modeling** command.
2. From the Select Modeling Method dialog box, under **Trade Area Analysis**, choose either the **Single Site** or the **Multiple Sites** option.
3. Click the **Next** button.
4. In the Select Table and Column dialog box, click the **Open Table** button to add a table to the Table to Grid list if the table of points you want to use is not listed.
5. From the **Select Table to Grid** list, choose the appropriate MapInfo table of points that contains the geographic location of each store/site.
6. From the **Attractiveness Column** list, choose the column that contains the attractiveness value for each site record. Store/site attractiveness is a relative number that can be defined by parameters such as total floor space or shelf space, number of parking spaces, age of the store, or any combination of elements which defines the appeal of an individual retail site.
7. For multiple sites, to use an unmapped MapInfo data file (an x, y, z file) that has not been converted to a vector point table (using the Create Points command in MapInfo Professional), do the following:
   - From the X Column list, choose the column containing the x-coordinates for each point.
   - From the Y Column list, choose the column containing the y-coordinates for each point.
   - Click the **Projection** button and choose the coordinate system of the location data.

8. In the **Data Description** box, type an annotation (maximum 31 characters). This will be carried as a header in the grid file.
9. Enable the **Ignore Records Containing Zero** check box if you want to include only non-zero records.
10. Click the **Next** button and enter the required information in the dialog box.
11. Click the **Finish** button.

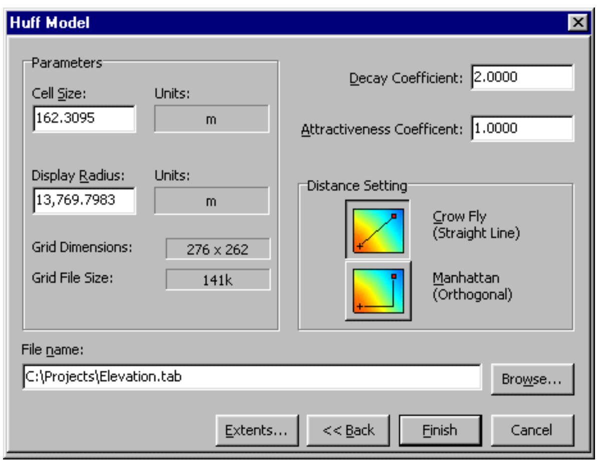
    If you are running the Patronage Probability For One Site option of the trade area analysis model, you are prompted to use the VM Picker Tool to select one site when you click the **Finish** button.

Once the grid is created, it appears in a Map window with a default colour palette applied. You can change the colour range assigned to the grid file. For more information, see **Using the Grid Colour Tool for Numeric Grids on page 97**.

Exploring the Huff Model Dialog Box

The Huff Model dialog box summarizes information relating to the site table and enables you to set parameters in the algorithm that control the creation of the trade area analysis grid. This dialog box is used for calculating trade areas for single and multiple sites

Note If you are solving a data set that spans a large area or if the data is entered far from the meridian or parallels of the projection, ensure the data set is in Latitude/Longitude coordinates. If you are working at the city level, there will be minimal error in the distance calculations if your data is in a projected map form and lies close to the central meridian or standard parallels of the projection.



The **Cell Size** box enables you to define the cell size in the native coordinate system units of the map. The grid dimension (in cell units) varies inversely with cell size: the smaller the cell, the larger the grid file. The value you choose should be a compromise between the degree of resolution required for analysis, the processing time, and file size. The grid dimensions (in cell width and height) and the grid file size (in kilobytes) are displayed and updated as the cell size is adjusted.

The **Display Radius** box indicates the radius around each store or site within which probability values will be calculated. The default display radius is set to equal 50 percent of the maximum diagonal extent of the entire data set.

The **Units** field displays the unit of horizontal distance measurement used in the model calculation (cell size and display radius). If the site map table is set to a Latitude/Longitude coordinate system, then the cell size unit is in degrees, and you can select and set the units for ***Display Radius*** in kilometres, miles, metres, or feet from the unit list box.

However, all distance calculations, when applied to the model, will automatically be converted to arc lengths corrected for earth curvature (oblate spheroid model). If the table is mapped to a defined projection in MapInfo Professional, then the ***Units*** field defaults to the distance unit set in MapInfo Professional's projection file (Pythagorean distance) and can be changed only if the point file is saved to a new projection.

The **Decay Coefficient** box enables you to enter an exponent that models the rate of decay in the drawing power of the sites as potential customers are placed farther and farther away from the site. Increasing the exponent will decrease the relative influence of the site on more distant customers. The optimal setting is two.

The **Attractiveness Coefficient** box enables you to enter an exponent to which the store attractiveness value is raised and enables you to account for non-linear behavior of the attractiveness variable. The default value is one, which assumes linear behaviour.

The **Distance Setting** section contains the following options:

- The **Crow Fly** (Straight Line) option enables you to measure the distance between the grid cell and the point using a straight line path.
- The **Manhattan** (Orthogonal) option enables you to measure the distance between the grid cell and the point location using the x and y difference between the two sites, that is, it combines the total x-axis and y-axis distance that must be covered to reach a destination. For Latitude/Longitude data, the distance is measured first travelling north-south then east-west. This option is provided to approximate distances when travelling on road networks that are oriented in a square grid pattern.

The **File name** box enables you to enter a new file name.

The **Extents** button displays a summary of the geographic size, the z-value range of the point database, and the data units.

Once the grid is created, it appears in a Map window with a default colour palette applied. You can change the colour range assigned to the grid file. For more information, see **Using the Grid Colour Tool for Numeric Grids on page 97**.

Suggested Reading on Trade Area Analysis

For a review covering the basics of probabilistic trade area analysis, refer to:

Dudley, G., 1996: *Probability Primer: Spatial Insights into StoreBase Trading Areas*. F.Y.I. Vol. 4, #2, August. A Publication of Blackburn/Polk Marketing Services Inc., Toronto.

For further readings on the topic of spatial interaction, refer to:

Fotheringham, S. and O'Kelly, M., 1989: *Spatial Interaction Models: Formulations and Applications*. Kluwer Academic Publishers.

Taylor, P. J., 1975: *Distance Decay Models in Spatial Interactions. Concepts and Techniques in Modern geography*. Norwich: Geo Books.

5

Creating Grids Using Other Methods

Vertical Mapper comes with a number of data preparation and grid creation methods that enable you to build grid files from data files and tables of regions, polylines, or points.

This chapter covers how to:

- import grids from other formats
- create point density grids (square area and kernel smoothing)
- convert a region to a grid
- buffer a grid
- use the Poly-to-Point tool to prepare data for creating a grid

In this chapter:

Creating Grids Using Other Methods

There are a number of ways you can create a grid other than by using the Interpolation or Modeling Wizards.

Import grids from other formats You can import a grid from any supported format. For more information on importing grids, see **Importing Grids on page 74**.

Create Point Density Grids Using the Point Density command from the Vertical Mapper, Create Grid menu, you can create a point density grid representing the density of events in an area. You can choose Point Density using Square Area or Point Density using Kernel Smoothing. For information about creating point density grids, see **Creating Point Density Grids on page 76**.

Convert a MapInfo region to a grid Using the Region to Grid command from the Vertical Mapper, Create Grid menu, you can convert a MapInfo region to a grid. For information about converting MapInfo regions, see **Region-to-Grid Conversion on page 80**.

Create a Buffer grid Using the Grid Buffer command from the Vertical Mapper, Create Grid menu, you can create a buffer around a line, polyline, point, or polygon. For information about grid buffering, see **Grid Buffering on page 81**.

Use the Poly-to-Point tool The Poly-to-Point tool does not create a grid, but it can be used to prepare line data for grid creation by extracting points from existing tables of polylines or regions. For information about the Poly-to-Point tool, see **Preparing Data using Poly-to-Point on page 84**.

Create grids by analyzing other grids Each of the tools used to analyze an existing grid creates a new grid with the results of the analysis. For more information about analyzing grids, see **Using Grids for Spatial Analysis on page 114**.

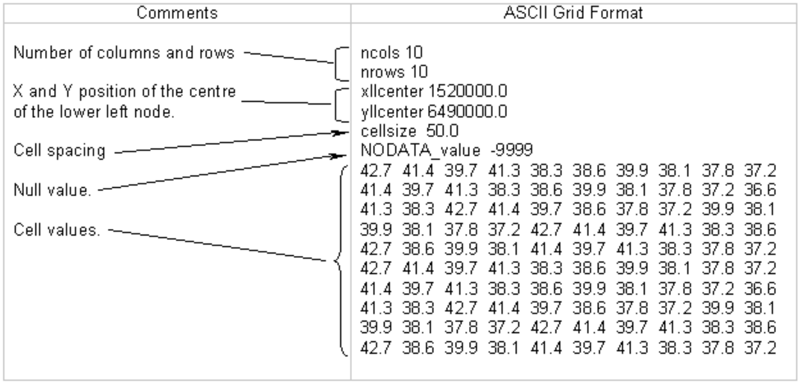
Grid Import Formats

Due to the ever-increasing usage of binary and ASCII-based grid files in desktop mapping environments, Vertical Mapper imports a number of formats for the more commonly used grid types. Most of these grids represent the elevation of a land surface, commonly referred to as Digital Terrain Models (DTMs) or Digital Elevation Models (DEMs).

The Grid Import utility can auto-detect and convert the following formats:

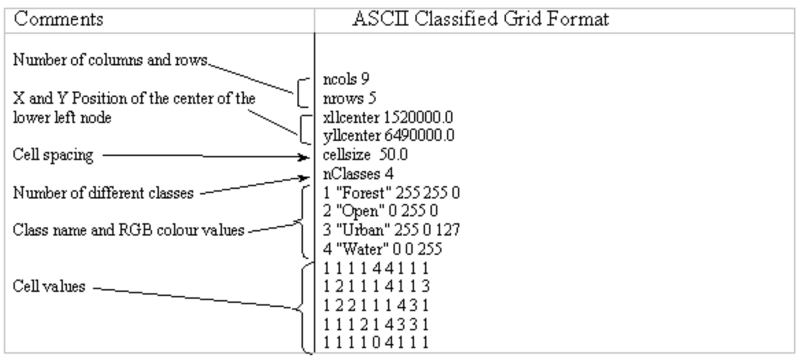
- ASCII Grid
- ASCII Classified Grid
- CRC-500
- Geological Survey of Canada Grid
- DTED
- GeoSoft
- MONA
- UK Ordnance Survey Grid
- USGS DEM
- SDTS - Spatial Data Transfer Standard Grid
- MIG Grid

ASCII Grid is a simple text-based format consisting of five lines of header information followed by rows of space-delimited float point values. Due to its simple and relatively compact nature, you should consider writing your data in this format when you want to import your own grid data into Vertical Mapper. When this file format is detected, a dialog box is displayed that enables you to set the projection, z-unit type, and a number used as a special flag indicating null value grid cells. The default null value is -9999.0; however, if the data includes -9999.0 as a valid numeric value, a new null can be assigned to grid cells with no value.



Example of a grid file illustrating the construction of an ASCII grid file.

ASCII Classified Grid is a simple text-based format consisting of general grid information in the file header followed by rows of space-delimited integer values. The header of the file contains a lookup table of class values, which define the integer representing each class name. The value 0 is used to define the null cell (a cell that is not defined in the lookup table). The lookup table should always begin sequentially with the value 1. The maximum number of classes included in this format is 65 535.



Example of a simple classified import file.

CRC-500 is a binary DEM for RF propagation modeling supported by Canada's Communications Research Centre. The grid cell resolution is 500 metres in a UTM projection using the NAD 27 datum for Canada.

DTED is a binary DEM originally supported by the U.S. Defence Mapping Agency (now NIMA). The Grid Import utility recognizes both Level 1 (3 arc-second) and Level 2 (1 arc-second) in the latitude/longitude coordinate system.

Geological Survey of Canada Grid is a text-based, delimited format grid file supported by the Survey for a number of geophysical data themes at a variety of grid cell resolutions and projections.

GeoSoft is a proprietary binary grid format supported by the GeoSoft Inc. suite of industry-standard geochemical and geophysical data interpretation applications. The format is extremely flexible but does not support any projection information in the header. The Grid Import utility prompts you to enter a MapInfo Professional-recognized coordinate system class. The Grid Import utility supports only uncompressed GeoSoft grids.

MONA is an x, y, z text-based, comma-delimited format that contains no header information. The data must be regularly spaced and sorted by row. The format is supported by the French government for European DEM coverage.

UK Ordnance Survey Grid is an ASCII non-delimited format supported by the Ordnance Survey with DEM coverage throughout the United Kingdom. It comprises a diverse series of format variations in which grid-based DEM Panorama and Profile products can be handled. A variety of resolutions are supported.

USGS DEM is an ASCII format DEM supported by USGS with coverage throughout the entire United States. The Grid Import utility automatically recognizes a variety of DEM coverages including the 7.5 minute, 30 arc-second, and 3 arc-second Latitude/Longitude grids, as well as UTM-projected orthogonal 30-metre grids.

SDTS is designed to store several different forms of spatial data such as vector and raster information. The Grid Import utility extracts only the grid information from the file.

MIG is the native data format of Pitney Bowes Software Inc. .

Importing Grids

A default file name will be created for each import file. If this file name conflicts with an existing one, you will be prompted to supply a new one. It is impossible to overwrite existing files while importing grids.

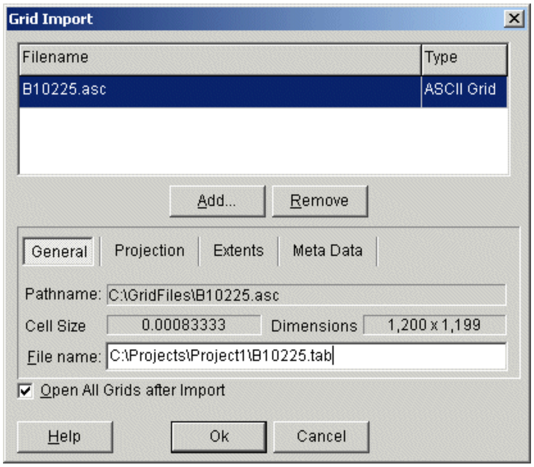
1. From the Vertical Mapper menu, choose the **Create Grid > Import Grid** command.
2. In the Grid Import dialog box, click the **Add** button.
3. In the Open dialog box, choose the file you want to import, and click the **Open** button.
4. In the Choose Projection For dialog box, choose the projection category and the category member, and click the **OK** button.
5. In the Enter Null Value For dialog box, define a null value or accept the default and click the **OK** button.

   The default represents a special value used in the file format to indicate “no data”.
6. In the Enter Z-Value For dialog box, type a z-value, and click the **OK** button.

   The Grid Import dialog box is updated with information attributed to the grid (for example, path name, cell size, projection, extents and meta data).
7. In the Grid Import dialog box, type a new file name in the **File name** box or accept the default.
8. Click the **OK** button.

Exploring the Grid Import Dialog Box

The top section of the Grid Import dialog box lists all the files you have chosen as well as the associated file formats. The bottom section presents four tabs containing information about the highlighted grid.



The **Add** and **Remove** buttons enable you to add or remove files from the Import Files list. When you choose a file and add it to the list, the Grid Import utility attempts to identify the file type that will be displayed next to the file name in the list.

The **General** tab enables you to identify the full path name of the import file, the cell size, and the estimated dimensions of the new grid file.

The **Projection** tab enables you to identify the coordinate system string that will be used. You can re-select your projection prior to import if required.

The **Extents** tab provides information concerning the bounding area of the new grid.

The **Meta Data** tab enables you to enter a description (up to 31 characters) for the grid. If the import file does not contain a z-unit description, you will be prompted on opening the file for import to choose an appropriate z-unit. You may change it at any point until you click the OK button. The null data value is a special value that will be used to indicate "no data". If this value is not provided by the import file, you will be prompted to enter it.

The **Open All Grids after Import** check box enables you to open all imported files after they have been imported.

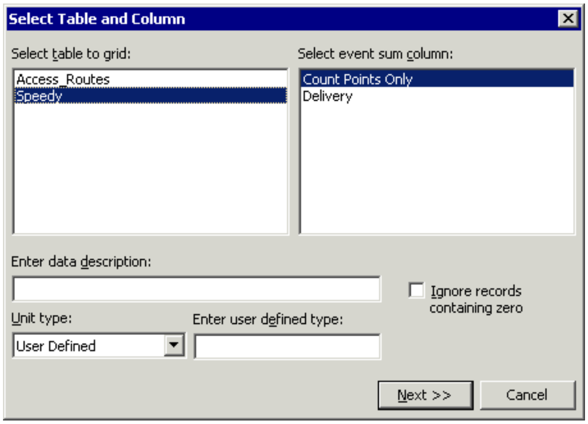
Creating Point Density Grids

Calculate Point Density techniques are used to calculate the density of events in an area. Vertical Mapper supports two techniques for calculating point density: Square Area and Kernel Smoothing. These options are available from the Vertical Mapper, Create Grid, Point Density menu.

Point Density using Square Area

This method of calculating point density results in a grid where each cell value represents the number of events within a user-defined area around the cell. For example, if the specified area is one square kilometre, then each cell will have the number of events within one square kilometre around the cell. Two methods of calculating or summing the number of events are provided. The first method counts the number of symbol objects found within the user-defined area. This method assumes that each symbol represents a single event. The second method is more sophisticated in that it allows for a single symbol to represent more than one event. It does this by accessing a column containing the number of events at each location in the database of the point file.

Exploring the Select Table and Column Dialog Box



This dialog box opens when you select either of the point density options. It enables you to select the table to grid and choose the Count Points Only option or specify a column containing the data to use.

The **Select Table to Grid** list displays a list of available tables.

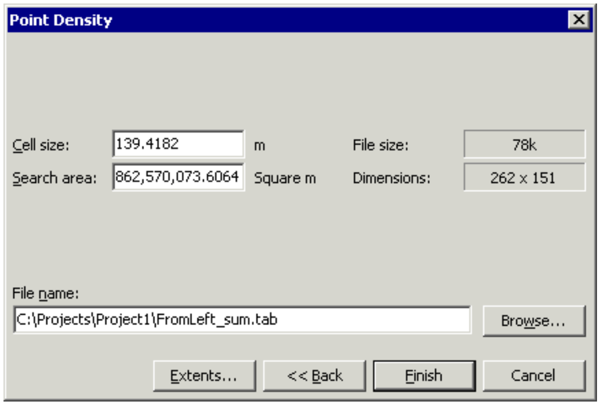
The **Select Event Sum Column** list displays “Counts Points Only” and a list of available columns to use; choose Count Points Only if you want only to count the points in the area; otherwise, choose the name of the column containing the data to use.

The **Enter Data Description** box enables you to enter a description for the data.

The **Unit Type** list enables you to choose the unit type. If you choose the User Defined option, you must specify the type in the User Defined Type box.

The **Ignore Records Containing Zero** check box when this check box is enabled, records containing zero are ignored.

Exploring the Point Density Dialog Box



The **Cell Size** box enables you to specify the cell size in the coordinate unit of the point file to be processed.

The **Search Area** box enables you to specify the area around each cell to search for events. Units are specified in the coordinate units of the point file to be processed.

Creating a Square Area Point Density Grid

1. From the Vertical Mapper menu, choose the **Create Grid, Point Density, Square Area** command.
2. From the Select Table to Grid list, choose the table you want to create a grid from.
3. In the Select Event Sum Column list, do one of the following:
   - If you want to count the points found inside the search area, choose Count Points Only.
   - If you want to sum the values contained in a given field in the database, choose the name of the column in the database that contains the values.
4. Click the **Next** button.
5. In the Point Density dialog box, enter the cell size and search area.
6. Click the **Finish** button.

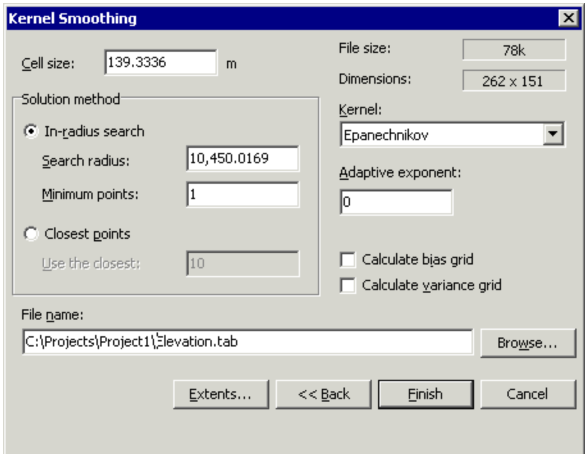
Point Density Using Kernel Smoothing

The Kernel Smoothing method enables you to take a set of discrete points, such as specified locations of crime or disease, and transform them into a continuous surface. These discrete points are known as “events”. By studying the events occurring within a specified radius, you can determine the probability of the event occurring in adjacent regions.

You can use two different variations of the Kernel Smoothing technique: weighted or classic. In the weighted variation, you can apply a weight to a point in order to reflect the occurrence of multiple events at a location. In the classic variation, all points are seen as single events and have the same weight.

Exploring the Kernel Smoothing Dialog Box

The Kernel Smoothing dialog box enables you to specify parameters specific to the Kernel Smoothing algorithm.

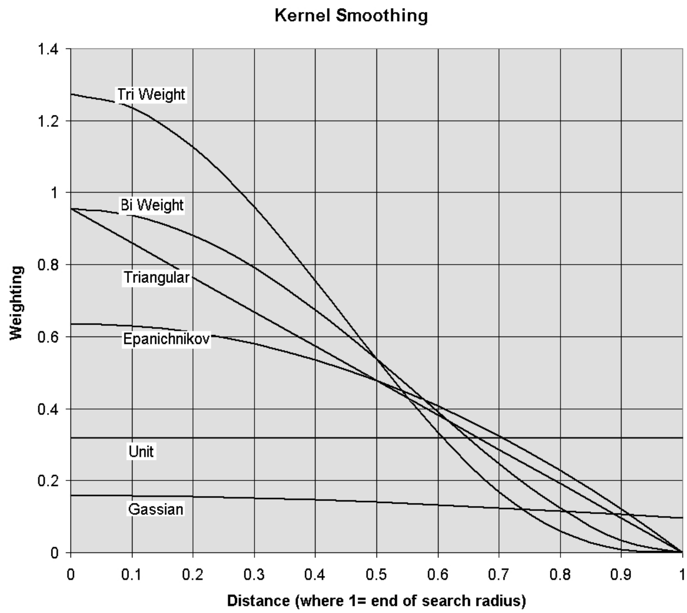


The **Cell Size** box enables you to specify the cell size in the coordinate unit of the point file to be processed.

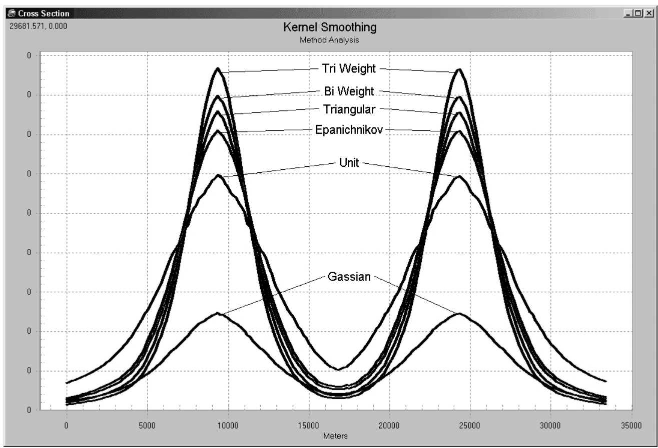
The **Solution Method** section contains the following options:

- The **In-Radius Search** option enables you to define the search radius and the minimum number of points within the search radius. If the number of points found is less than the minimum value, a null value is returned. The size of the search radius depends on the type of data you are analyzing.
- The **Closest Points** option enables you to define the number of closest points to include in the calculation. This option is useful if the quality of your data is in doubt.

The **Kernel** list enables you to apply one of six kernel estimates to the point file: Unit, Gaussian, Triangular, Epanechnikov, Bi-Weight, and Tri-Weight. The difference between them is the degree of weighting points have with respect to their distance to the grid.



This figure illustrates the weighting of each of the six kernel estimate options.



This figure illustrates the results of using each type of weighting option; the choice affects the "pointiness" of the grid in high density areas.

The **Adaptive Exponent** box enables you to specify the coefficient to adjust the kernel algorithm to the peculiarity of the data set. The value you specify must be between 0 and 1.

The **Calculate Bias Grid** and **Calculate Variance Grid** check boxes enable you to generate a bias grid and a variance grid.

The **Extents** button displays a summary of the geographic size and the z-value range of the original point database, the density of the points, and the data value units.

Region-to-Grid Conversion

The Region-to-Grid conversion process involves extracting a text or numeric value from a column in the region table and assigning this value to all the grid cells that fall inside that region. If the assigned value taken from the MapInfo table is a text string, the process automatically creates a classified grid file. If the assigned value is numeric, it generates a numeric grid file. In both cases, the resulting grid appears similar to the original region map. However, it often tends to look somewhat “notchy” or pixelated. For classified grids, the process assigns colour to a grid cell when it extracts an attribute from the MapInfo region table.

Once you have converted a region table to a grid, you can analyze the data using processes that cannot be applied to standard MapInfo vector regions. For example, using the Grid Query command accessible from the Analysis button in the Grid Manager, classified geographic information converted to grid format can be overlain with numeric grid data and queried using standard Boolean operators to create a new derivative grid map that meets the conditions of the query statement. Performing powerful spatial analysis routines using any type of geographic data is the single greatest advantage afforded by raster grid.

Converting a Table of MapInfo Regions to a Grid

The Region to Grid dialog box is used to convert a table of MapInfo regions to a numeric or classified grid format.

1. From the Vertical Mapper menu, choose the **Create Grid > Region to Grid** command.
2. In the Region to Grid dialog box, from the Source Table list, choose the MapInfo region file that contains the information to be converted.
3. From the Column list, choose a column that contains the attribute information to be assigned to the grid cells.

   If you choose a column containing numeric data, you generate a numeric grid. If you choose a column containing text data, you generate a classified grid.
4. In the **Cell Size** box, type the grid cell size for the new grid.

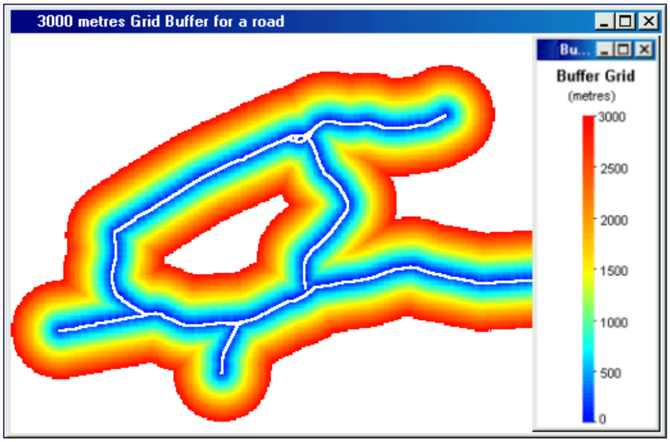
   The conversion process builds a new grid file that, by default, covers the entire geographic extent of the MapInfo region table. The grid file size and the grid dimensions will therefore increase as the cell size decreases.
5. In the **Data Description** box, assign an annotation (maximum 31 characters) that will be carried as a header in the grid file.
6. In the **File Name** box, type a new file name or accept the default.
7. Click the **OK** button.

Once the grid file is created, it appears in a Map window with the same colours assigned to each grid cell, if any, that were assigned to the MapInfo regions. You can change the colours assigned to the grid using the Colour tool accessible from the Grid Manager. For more information, see **Using the Grid Colour Tool for Numeric Grids on page 97** or **Using the Dictionary Editor for Classified Grids on page 99**.

Grid Buffering

In a vector GIS applications such as MapInfo Professional, you can create regions around map objects at a predefined distance. You cannot, however, determine distances within a buffer region. For example, if a highway is buffered at a distance of one kilometre, you will not be able to determine the precise distances that fall within the buffered area (for example, 750 m or 300 m). In Vertical Mapper, the value of each cell in a buffer grid is calculated as the distance to the closest input object. As a result, you can determine precise distances within the buffered region.

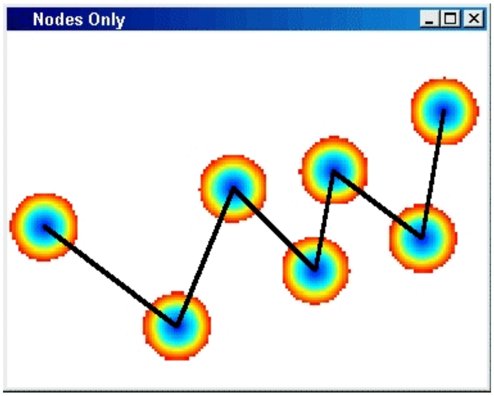
Input objects supported by this tool are line, polyline, point, and polygon objects. Polygon objects are treated as polylines, where the buffers are created using the object's boundary, and values are calculated on either side of this boundary (inside and outside of the polygon).



The Grid Buffer tool is used here to build a buffer grid of continuous distance from a road network.

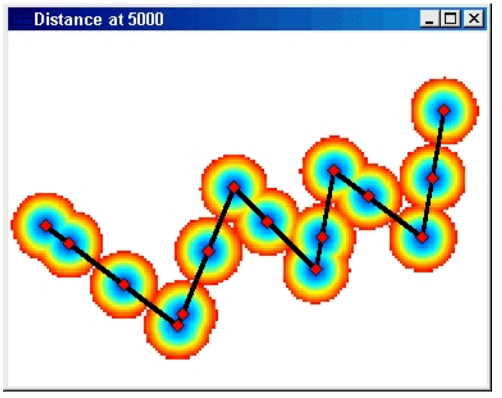
You can choose one of two options in the Grid Buffer tool: Use Nodes Only and Use Maximum Distance.

As shown in the next figure, when you choose the Use Nodes Only option, buffers are created only at the junction of the line's segments but at no other places.



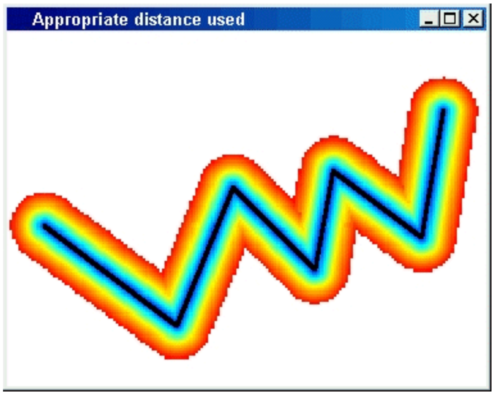
This graph shows the result when the Use Nodes Only option is chosen.

If you choose the Use Maximum Distance setting, you must specify the maximum distance between locations where a circular buffer will appear. If the maximum distance setting is too great, as shown in the next figure, there will not be enough circular buffers to cover the length of the line.



This graph shows the result when the Use Maximum Distance option is chosen and the distance setting is too big.

By reducing the distance setting, the buffers will cover the whole line. This is illustrated in the next figure. Generally, the distance setting should be at least half of the buffer distance. For example, if you create buffers at 1 000 m, the maximum distance setting should be at least 500 m.



This graph shows that the buffer covers the whole line when the maximum distance is reduced properly.

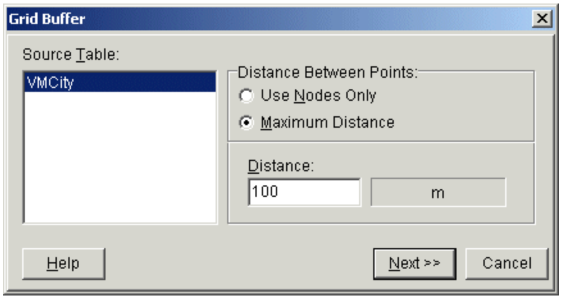
Creating a Grid using the Grid Buffer Tool

1. From the Vertical Mapper menu, choose the **Create Grid > Grid Buffer** command.
2. From the Source Table list, choose a table for processing.
3. In the **Distance Between Points** section, choose either the **Use Nodes Only** option or the **Use Maximum Distance** option and define the distance value between points in the **Distance** box.

   The maximum distance between points should always be equal to or less than the search radius.
4. Click the **Next** button.
5. Define **Cell Size** and **Buffer Distance** settings.

6. In the **Enter Data Description** box, assign an annotation (maximum 31 characters) that will be carried as a header in the grid file.
7. In the **Output Grid Name** box, type a new file name or accept the default.
8. Click the **Finish** button.

Exploring the Grid Buffer Dialog Boxes

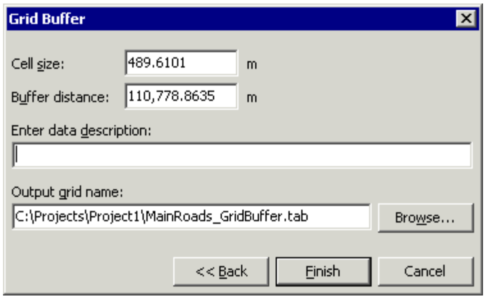


The **Source Table** list enables you to choose from tables available for processing.

The **Use Nodes Only** option enables you to create points from the existing nodes along the polylines.

The **Maximum Distance** option enables you to add extra points along polylines ensuring that no two points are farther apart than the Distance value.

The **Distance** box enables you to enter the maximum distance between points.



The **Cell Size** box enables you to define the cell size in the native coordinate system units of the table. The grid dimension (in cell units) varies inversely with cell size: the smaller the cell, the larger the grid file. The value you choose should be a compromise between the degree of resolution required for analysis and visualization purposes, the processing time, and the file size. The units are displayed next to the boxes. If the source polyline file is in a latitude/longitude projection, the units of the radius value are displayed. Otherwise, the units are those of the table's coordinate system.

The **Buffer Distance** box enables you to define the maximum size, in radius units, of a circular zone centred on each grid node. For each point lying within this search zone, the distance between the grid cell (the centre of the zone) and the point is measured. The value of the grid cell will be the distance to the closest point. If no point is within the search radius, the cell will be assigned a null value.

The **Enter Data Description** box enables you to assign an annotation (maximum 31 characters) that will be carried as a header in the grid file.

The **Output Grid Name** box enables you to assign a file name to the points file you create through the Grid Buffer technique.

Preparing Data using Poly-to-Point

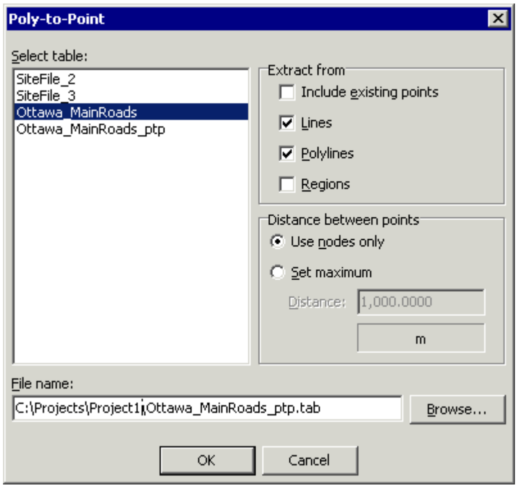
Using Poly-to-Point, you can extract points from existing tables of lines, polylines, or regions to prepare data for creating grids.

1. From the Vertical Mapper menu, choose the **Create Grid > Poly-to-Point** command.
2. From the Select Table list, choose a table for processing.
3. In the **Extract From** section, enable any or all of the **Include Existing Points, Lines, Polylines,** and **Regions** check boxes to determine the type of objects in the table from which points are to be extracted.
4. In the **Distance Between Points** section, choose either the **Use Nodes Only** option or the **Set Maximum** option and define the distance value between points in the **Distance** box.

   The maximum distance between points should always be equal to or smaller than the search radius.
5. In the **File Name** box, type a new file name or accept the default.
6. Click **OK**.

   Once the point table is created, an information dialog box appears with a summary of the number of objects (points, polylines, or regions) processed and the total number of points created.

Exploring the Poly-to-Point Dialog Box



The **Select Table** list enables you to select a table for processing.

The **Extract From** section enables you to choose any or all of the **Include Existing Points, Lines, Polylines**, and **Regions** options to determine the type of objects in the table from which points are to be extracted.

The **Distance Between Points** section includes the following options:

- The **Use Nodes Only** option enables you to specify that only the nodes are included in the calculation.
- The **Set Maximum** option enables you to define the distance value between points in the Distance box.

6

Working with Grids

Gridding is the basic mapping technique used in Vertical Mapper. The Grid Manager is the central dialog box from which you can open, sort, view, and manipulate grids. Vertical Mapper also includes several tools that enable you to extract information from grids, create legends for grids, display graphs relating to grid information, and create contour grids.

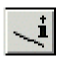
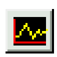
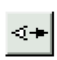
This chapter covers how to:

- use the Vertical Mapper toolbar
- use the Grid Manager
- use the Grid Info, Region Info, and Line Info tools
- use the Grid Colour Tool and the Dictionary Editor
- display legends
- work with graphs
- create polyline and region contour grids

In this chapter:

Working with the Vertical Mapper Toolbar

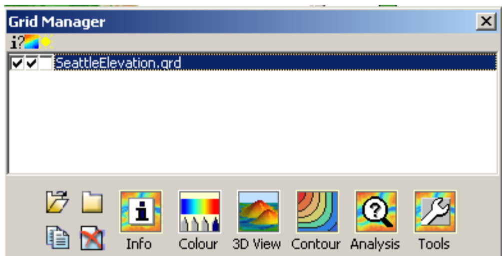
The Vertical Mapper toolbar provides you with easy access to a number of tools. Using these tools, you can show/hide the Grid Manager, get information about various grids, and conduct a variety of grid analyses. If the Vertical Mapper toolbar is not visible, you can display it using the **Options > Toolbars** command.

Toolbar button	Functionality
	The Show/Hide Grid Manager button enables you to show or hide the Grid Manager.
	The Info button enables you to show and capture grid information for selected points in one or more active grids. For more information, see **Using the Grid Info Tool on page 94**.
	The line info button enables you to show grid information for a polyline. For more information, see **Using the Line Info Tool on page 96**.
	The region info button enables you to show grid information for a region. For more information, see **Using the Region Info Tool on page 95**.
	The Cross Section button enables you to conduct cross section analysis of grids. For more information, see **Creating a Cross Section on page 125**.
	The Grid Trim button enables you to trim grids. For more information, see **Trimming a Grid on page 150**.
	The Viewshed Analysis button enables you to conduct Viewshed analysis on grids. For more information, see **The Viewshed Function on page 132**.
	The Point-to-Point Visibility button enables you to conduct Point-to-Point visibility analysis on grids. For more information, see **The Point-to-Point Visibility Function on page 129**.
	The VM Picker Tool button enables you to select objects in a Map window. This button is only enabled when you are required to select an object in a Map Window for a Vertical Mapper procedure. Uses include picking a site for single site trade area analysis and picking boundary regions for Voronoi or Natural Neighbour Analysis.
	The Create Drape File button enables you to create 3D drape files. For more information, see **Making 3D Drape Files on page 201**.

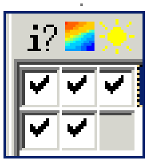
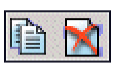
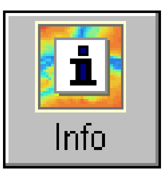
Working with the Grid Manager

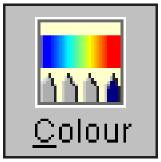
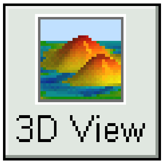
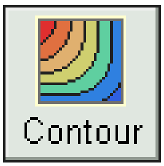
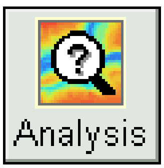
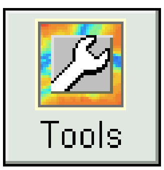
The Grid Manager enables you to obtain information about grid files, access the Grid Colour tool or the Dictionary Editor, run the 3D Viewer, create a 3D drape file, create region and polyline contours, and analyze or modify your grid files.

- On the Vertical Mapper Project toolbar, click the **Show Grid Manager** button  .



Note You can also open the Grid Manager by choosing the Show Grid Manager command from the Vertical Mapper menu.

Grid Manager Options	Description
	Three check boxes are displayed next to the grid name. By enabling the Active check box you can make a grid active and query it using the Grid Info tool. By clearing the Colour Gradient check box, you can turn off the colour profile for a grid and display it in grey-scale if relief shading has been applied. By enabling the Relief Shading check box, you can turn on the relief shading function if it has been previously generated for the numeric grid using the Grid Colour tool.
	The Open Grid and Close Grid buttons enable you to open or close grid files.
	The Copy Grid As and Delete buttons enable you to copy or delete grid files.
	The Grid Info button expands the grid manager to provide information on a selected grid in terms of geographic characteristics, z-units, meta data, legend, and histogram.

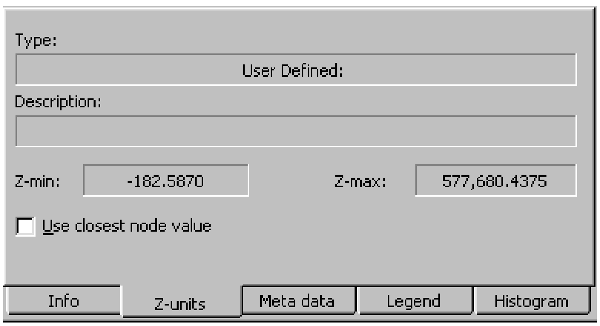
Grid Manager Options	Description
	The Grid Colour Tool button enables you to access the Grid Colour Tool for numeric grids (for example, elevation .grd files) and the Dictionary Editor for classified grids (for example, clutter .grc files).
	The Grid View button enables you to make 3D drape files or to run the 3D Viewer. For more information, see **Launching GridView on page 188**.
	The Contour Grid button enables you to generate Region and Polyline contours for numeric grids. For classified grids, it creates region contours only. For more information, see **Contouring with Grids on page 110**.
	The Analyze Grid button enables you to analyze grid files using the calculator or the grid query command, to create slope and aspect grids, or to create a cross section. You can also perform a point, line, or region inspection, calculate point-to-point intervisibility, and perform a viewshed analysis. For more information, see **Using Grids for Spatial Analysis on page 114**.
	The Grid Tools button enables you to alter meta data, filter classified grid files, export grid files, create grid legends, and reclass or reproject files. You can also resize, splice, and trim grid files and launch the Classified Grid Filter tool. For more information, see **Using the Grid Tools on page 138**.

Open grids are listed in the window at the top of the Grid Manager. You can make a grid active, or apply colour gradients or relief shading, by enabling the appropriate check box next to a grid. You can also change the order in which grids are listed in the Grid Manager by choosing a grid and dragging it to a new position in the list.

When you click the Grid Info button, the Grid Manager expands to display a series of tabs that provide information on each selected grid file including the following information:

- geographic characteristics
- z-unit characteristics
- lineage and history (meta data)
- data legend
- grid histogram for both numeric and classified grids

The **Info** tab summarizes the spatial characteristics of the selected grid including cell size, geographic unit of measure, grid dimension (in cells), file size, x and y extent of the grid, light source settings that define the relief shading of the grid (if applicable), and the name of the MapInfo Professional coordinate system to which the grid is referenced. Light source information does not apply to classified grids.

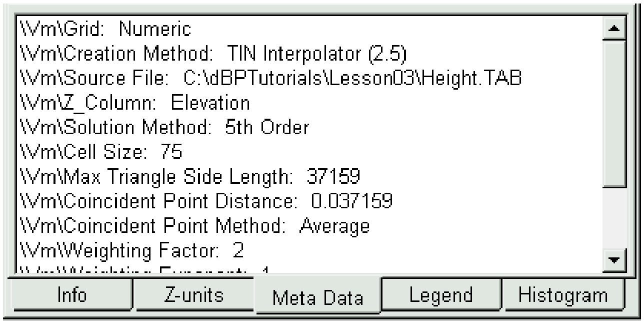


The **Z-units** tab provides a summary of the grid data characteristics including the data value range, the data type, and data description. The data type and description are entered when the grids are built and can also be changed using the Alter Meta Data command.

If the **Use Closest Node Value** check box is enabled, the value of the closest node is returned whenever the grid is queried. Consequently, the value of any location within a cell will be the same. If the check box is cleared, the values of adjacent cells are used to estimate the value of locations inside a cell, and values of individual locations within the cell will be different. This setting affects all of the following functions:

- grid calculations
- grid queries
- point-to-point visibility
- cross section

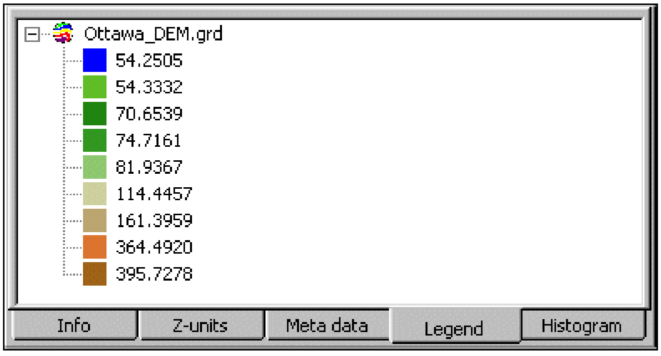
- point inspection
- line inspection
- grid info
- line info



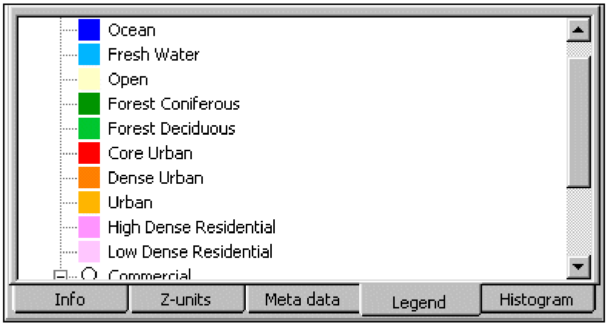
The **Meta Data** tab provides access to historical information relating to the creation of the grid and is carried as a header string within the grid file. It includes important information such as the name of the original data file from which the grid was derived, the method of grid interpolation used to create it, including the interpolation settings, a date and time stamp of the original build, and the identity of the builder (if available). Additional information can be appended using the Alter Meta Data command accessible from the Tools menu in the Grid Manager. For more information, see **Using the Alter Meta Data Tool on page 138**.

Note If only one grid has been opened in MapInfo Professional, you may have to click on that grid in the Grid Manager to view the meta data.

The **Legend** tab displays the colour inflection values that define the gradient colour scheme of the grid. These are the same values that are assigned in the Grid Colour Tool and appear as defaults in the Legend Generator. If you query a classified grid file, the legend window displays the default colour values applied to each class found in the grid.

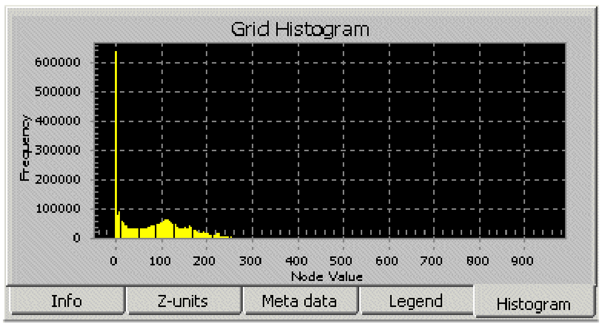


Legend for a numeric grid.

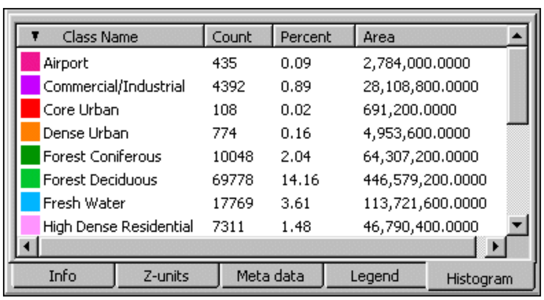


Legend for a classified grid.

The **Histogram** tab displays a histogram of the grid data in a graph window. The histogram provides you with a visual display of the distribution of your data so that you can make better decisions for grid display and grid analysis operations. This window can also display classified grid data by class name, class count, percentage, and area. You can customize the display of the histogram by right-clicking on the histogram.



Histogram for a numeric grid



Class Name	Count	Percent	Area
Airport	435	0.09	2,784,000.0000
Commercial/Industrial	4392	0.89	28,108,800.0000
Core Urban	108	0.02	691,200.0000
Dense Urban	774	0.16	4,953,600.0000
Forest Coniferous	10048	2.04	64,307,200.0000
Forest Deciduous	69778	14.16	446,579,200.0000
Fresh Water	17769	3.61	113,721,600.0000
High Dense Residential	7311	1.48	46,790,400.0000

Histogram for a classified grid

Using the Grid Info Tool

The Grid Info tool enables you to query and return grid values from selected locations in the grid file and view the results in a Grid Info dialog box. If more than one grid is open and active, values will be returned from each one.

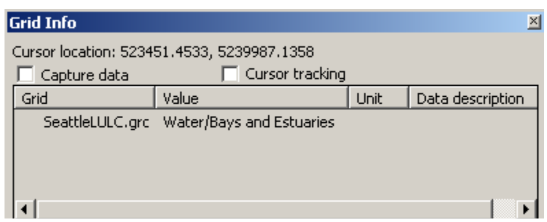
Only grid information from those grids that are active in the Grid Manager and are in the same geographic location as the point being inspected are displayed. The grid files being inspected do not have to be layers in the current Map window. The Grid Info tool simply examines the x- and y-coordinates of the location being inspected and returns the information from those active grids in the Grid Manager that are geographically coincident. If you need to inspect a large number of predefined points, you can use the Point Inspection function. For more information, see **The Point Inspection Function on page 126**.

1. On the Vertical Mapper toolbar, click the **Grid Info** button. 
2. Click in the Map window.

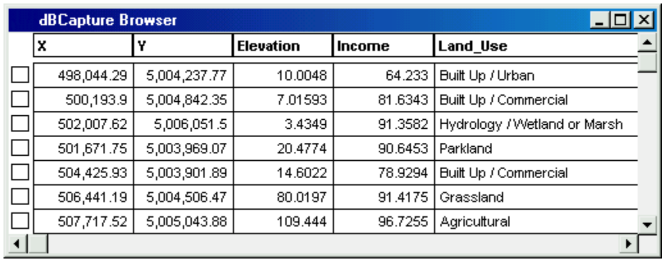
Note You can choose whether or not values are interpolated within cells by enabling or clearing the Use Closest Node Value check box on the z-units tab in the Grid Manager. For more information, see **Working with the Grid Manager on page 89**.

Exploring the Grid Info Dialog Box

The Grid Info dialog box displays the cursor location as well as information about all open and active grids at that location, including the grid file name, grid cell value, the data unit (if available), and the data description (if available). You can sort the information either alphabetically or numerically, in ascending or descending order, by clicking the button with the column name Grid, Value, Unit, or Data Description. When sorting, an arrow appears that indicates the direction of the sort. When the arrow is not visible, the grids will be placed in the same order as they appear in the Grid Manager. You can resize a column to the length of the longest item in that column by double-clicking on the divider to the right of the column.



The **Capture Data** check box enables you to create a new MapInfo table called vmCapture to automatically store the results of all inspections. vmCapture is automatically opened in a Browser window when you choose this option.



X	Y	Elevation	Income	Land_Use
498,044.29	5,004,237.77	10.0048	64.233	Built Up / Urban
500,193.9	5,004,842.35	7.01593	81.6343	Built Up / Commercial
502,007.62	5,006,051.5	3.4349	91.3582	Hydrology / Wetland or Marsh
501,671.75	5,003,969.07	20.4774	90.6453	Parkland
504,425.93	5,003,901.89	14.6022	78.9294	Built Up / Commercial
506,441.19	5,004,506.47	80.0197	91.4175	Grassland
507,717.52	5,005,043.88	109.444	96.7255	Agricultural

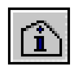
Every capture session is entered into the vmCapture table and, therefore, you cannot make any changes using the Grid Manager during a session without saving the current vmCapture table. Changes include adding or removing other grid files or making existing ones active or inactive. Vertical Mapper always prompts you to save the current vmCapture table to another name.

The **Cursor Tracking** check box enables you to continually update the Grid Info dialog box with values from the active grids in the Grid Manager by moving the mouse cursor.

You cannot use the Capture Data and the Cursor Tracking options at the same time.

Using the Region Info Tool

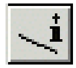
The Region Info tool displays a statistical summary of the data within a selected region. The values displayed are for the grid highlighted in the Grid Manager. To display statistical information for a large number of regions contained in the same table, use the Region Inspection function. For more information about region inspection, see The Region Inspection Function on page 128.

1. On the **Vertical Mapper** toolbar, click the **Region Info** button .
2. Select a region in the Map window.

Note The Volume calculation is the sum of all the values found in the region multiplied by the cell dimensions. This value is only meaningful when the z-unit is a linear measurement. The % null value indicates how much of the enclosed area contains null values.

Using the Line Info Tool

The Line Info tool displays a statistical summary of the data along a selected line. The values displayed are for the grid highlighted in the Grid Manager. To display information for a large number of lines contained in the same table, use the Line inspection tool. For more information about line inspection, see **The Line Inspection Function on page 126**.

1. On the Vertical Mapper toolbar, click the **Line Info** button .
2. Click on a line in the Map window.

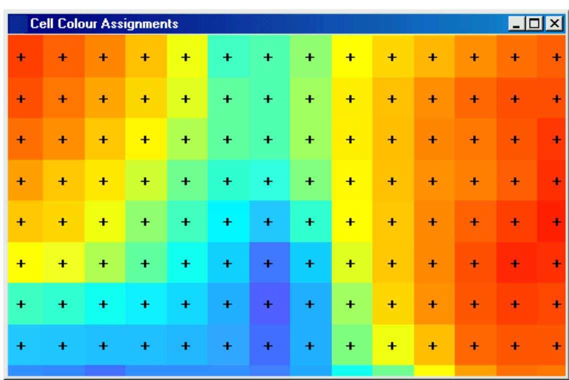
Note The number of samples taken along the line is determined by the Cross Section: No of Samples parameter in the Preferences settings. The default is 100. To change this value, choose the **Vertical Mapper > Preferences** command and enter a new number in the No. of Samples box. For more information about preferences, see **Setting your Preferences on page 204**.

You can choose whether or not values are interpolated within cells by enabling or clearing the check box on the z-units tab in the Grid Manager. For more information, see **Working with the Grid Manager on page 89**.

Using Colour in Grids

The use of colour is an effective way to give meaning to large stores of data. The display of grid data in Vertical Mapper is achieved by assigning a range of colour values, defined by a series of colour inflection points, to each grid cell based on the numeric or character value assigned to it.

For numeric grids, the colour inflection points are carried in the header portion of the grid file and can be easily modified using the Grid Colour Tool, which is a powerful tool used to present data visually. When a grid file is opened, the colour information in the header is read, and the colour profile is assigned to the grid cells and displayed in the Map window. In this manner, the image displayed on-screen is a true representation of the grid file, where each cell of the image represents a grid cell centred on the grid node as illustrated in the next figure.



Each grid cell is assigned a colour value as defined in the header portion of the file.

Any numeric grid file generated in Vertical Mapper, using interpolation or modeling, is automatically assigned a default colour palette that is applied when the file is first opened.

Colour is assigned to a numeric grid file according to ranges of grid values. Assume that a grid consists of values from zero to 100. Discrete colour gradients are then assigned by range according to defined inflection points. For example, the range from zero to 25 would be assigned colours varying from dark blue for the lowest value to medium blue for the highest value, while the range from 26 to 50 would be assigned colours varying from medium blue for the lowest value through green to yellow for the highest value in the range, and so forth. This technique is a standard method of presenting quantitative data in a continuous gridded surface using a gradient colour model, where blue is commonly used to represent low or “cold” values, and red is commonly used to represent high or “hot” values.

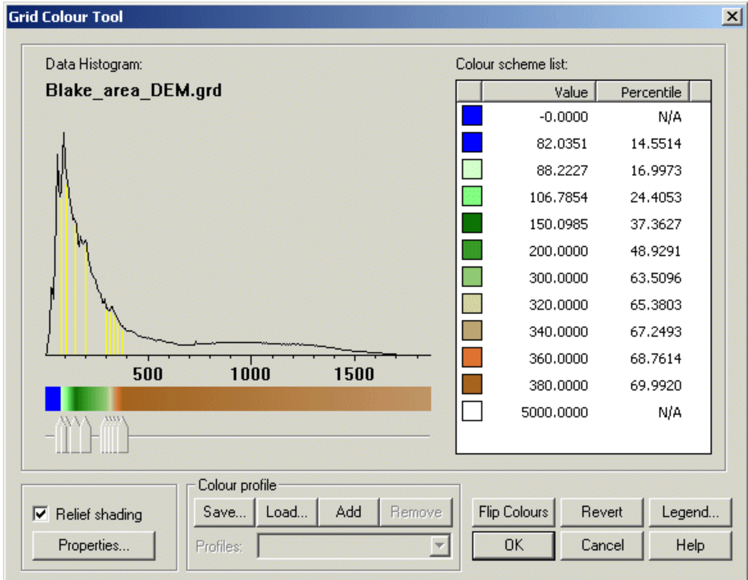
In addition to assigning colour gradients to numeric grids, you can also apply variations in light and shadow. Relief shading is an effective visualization process that creates 3D-like images from a continuous grid surface. The procedure calculates light and shadow effects based upon the angle of incidence of an imaginary point source of light to create grey-scale or colour images.

Vertical Mapper also supports the generation of true 3D perspective views using GridView. This tool gives you the ability to compose a Map window of any vector, raster, or textual entity that coincides with the spatial extents of the grid file and then drape the window onto a 3D representation of the grid surface. Perspective 3D rendering is supported in Vertical Mapper through implementation of the OpenGL graphics language.

In assigning colour to classified grids, Vertical Mapper uses a separate function called the Dictionary Editor. Using this tool, you can assign unique colours to each category of grid cells.

Using the Grid Colour Tool for Numeric Grids

All the display and colour options available for numeric grid files are contained in the Grid Colour Tool. The settings available in the Grid Colour Tool allow extremely precise control of gradient colour spread across the range of values found in the grid file. Discrete colour gradients are assigned by range according to defined inflection points. The Grid Colour Tool enables you to place up to 64 inflection points. A histogram showing the distribution of values in the grid file is displayed in the Grid Colour Tool window to assist in the placement of colour inflection points.



1. In the **Grid Manager**, choose a numeric grid (.grd).
2. Click the **Colour** button 

.
3. Do any of the following:
   - To add a colour inflection point, double-click the colour slider bar.
   - To delete a colour inflection point, click a colour inflection point to highlight it and press the Delete key.
   - To define a new colour for the inflection point, double-click on a colour inflection point.
   - To move an inflection point, click a colour inflection point and move the cursor while holding down the left mouse button. This will update the value for this inflection point in the Colour Scheme list.

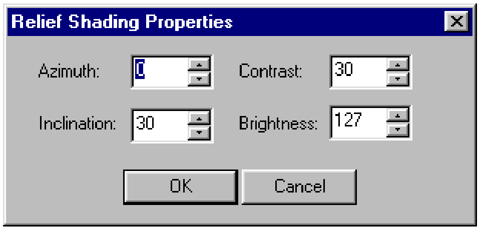
     The calculated values in the **Value/Percentile** list are automatically updated.
   - To reverse the colour inflection points, placing the inflection points in reverse order and updating the gradient colour pattern, click the **Flip Colours** button.
   - To change colour values and percentiles, click an entry in the **Value/Percentile** list to make the value editable and then enter a new value between the values above and below it. This will move the inflection point to the appropriate location on the colour ramp.
   - To restore original colour values and percentiles of the open grid in the current session, click the **Revert** button.
   - To create a legend for the grid, click the **Legend** button.
4. In the **Colour Profile** section, do one of the following:
   - Click the **Save** button to save colour settings as a text file with a .vcp extension.
   - Click the **Load** button to load a .vcp file.
   - Click the **Add** button to add the current profile to the registry.
   - Click the **Remove** button to remove the selected profile from the registry.
5. If you want to redefine the grid colours based upon how they would be illuminated by a single light source, enable the **Shading** check box and click the **Properties** button.

6. Click the **OK** button.

Note In deciding whether to save colour inflection points by value or by percentile, use the following guidelines. If it is more important to assign specific colours to specific values in a series of related grid files, then save by value. If it is more important to assign a particular colour range to a series of related grid files where the value range may vary considerably, then save by percentile.

Exploring the Relief Shading Properties Dialog Box

The Relief Shading Properties dialog box controls the basic elements of grey level adjustment using a Hue/Saturation/Luminescence (HSL) colour model. Grids can be viewed in grey-scale only when relief shading has been applied.



The **Azimuth** box enables you to set the direction (in degrees) from which the imaginary light source emanates.

The **Inclination** box enables you to set the angle of incidence (in degrees) of the imaginary light source measured from a horizontal plane. The effect of using different values depends upon the general “topography” of the grid surface.

The **Contrast** box enables you to determine the difference between the lightest and darkest tones in an image. Increasing the contrast setting will make the brighter areas brighter and the darker areas darker. Both are measured on a scale of one to 255, and the default values presented in the dialog box are considered appropriate for most renderings.

The **Brightness** box enables you to set a measure of both the colour intensity and overall greyness of an image. Increasing brightness will increase colour intensity as well as reduce the greyness of an image.

Note Setting the azimuth to 90 through 270 degrees may cause an optical illusion where hills look like valleys and valleys look like hills.

Using the Dictionary Editor for Classified Grids

The Dictionary Editor is used to assign colour values to each unique category of cells found in a classified grid, just as the Grid Colour tool is used to assign gradient colours according to the range of values found in a numeric grid.

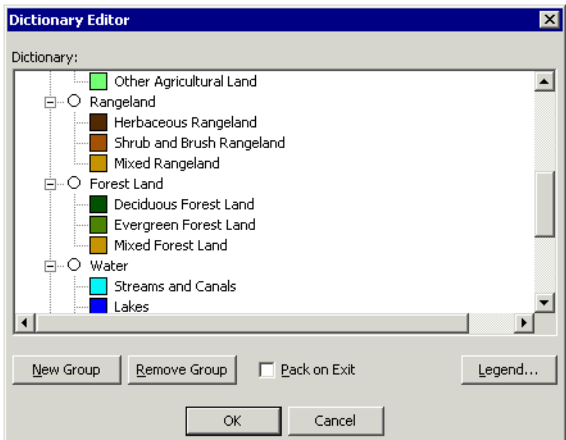
1. In the **Grid Manager**, choose a classified grid (.grc).

2. Click the **Colour** button. 

Exploring the Dictionary Editor Dialog Box

All of the display and colour options available for classified grid files are contained in the Dictionary Editor.

This dialog box displays a complete hierarchical list of all of the groups and subgroups of classified cells that comprise the grid file as well as the corresponding colour assigned to each unique category.



You can move a classification entry by choosing it and dragging it to a new position. You can edit the colours assigned to any category by double-clicking on an entry.

The **New Group** button enables you to add a new group to the dictionary. You can also access this command by right-clicking and choosing the New Group command from the shortcut menu.

The **Remove Group** button enables you to delete a group from the dictionary. You can also access this by right-clicking and choosing the Remove Group command from the shortcut menu.

The **Pack on Exit** check box enables you to re-number the internal class number for all classes starting sequentially at one. For grids that have fewer than 255 classes, this may reduce the size of the grid file.

The **Legend** button enables you to specify range, text, and format options for the legend.

Displaying Legends

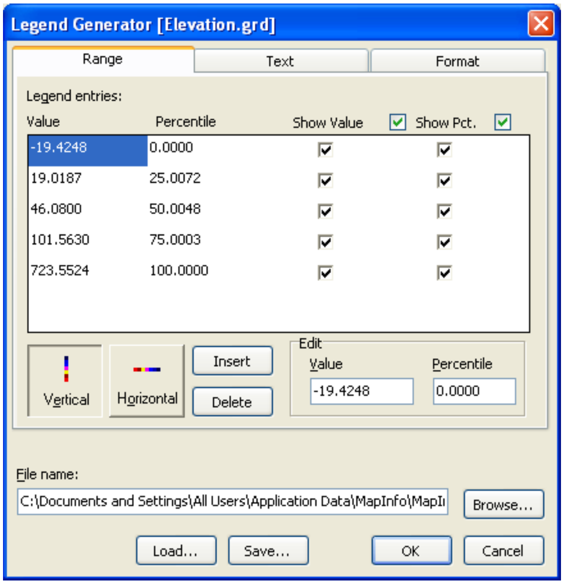
In order to understand the visual importance of a grid, you can create a legend for it.

1. From either the **Grid Colour Tool** for numeric grids or the **Dictionary Editor** for classified grids, click the **Legend** button.
2. In the **Legend Generator**, define the range, text, and format options.
3. Click the **OK** button to display the legend in a new Map window.

Note You can also access the Legend Generator from the Tools button in the Grid Manager. Choose the Grid Legends command, click the Ellipsis button next to the grid name, and then click the Legend button.

Exploring the Legend Generator Dialog Box

This dialog box provides options for constructing a legend showing either the continuous range of colours and values that comprise a numeric grid or the colours and class names contained in a classified grid.



The **Range** tab displays and allows editing of the values to be plotted on a legend bar. The values and percentiles that appear are those that you have defined in the Grid Colour Tool as the colour inflection points.

For numeric grids the values and percentiles that appear in the top portion of the dialog box represent the colour inflection points currently used. It is these values that will be used by default. For classified grids this area displays only the different class names.

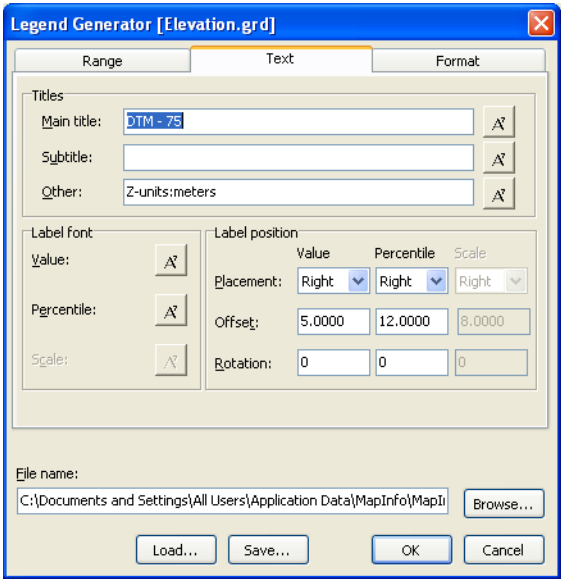
For numeric grids, you can enter new values into the legend using the Insert button or you can remove values using the **Delete** button. These commands are not available for classified grids.

The following options are available on the Range tab:

The **Show Value** check box and the **Show Pct**. check box enable you to display values only, percentiles only, or both beside the legend colour bar. These check boxes are not available for classified grids.

The **Horizontal** or **Vertical** buttons enable you to set the orientation of the colour legend. Classified grids will appear only in vertical form.

The **Text** tab enables you to control the placement and formatting of textual elements that appear in the legend including the title, label position, font style, and size.



The following options are available on the **Text** tab:

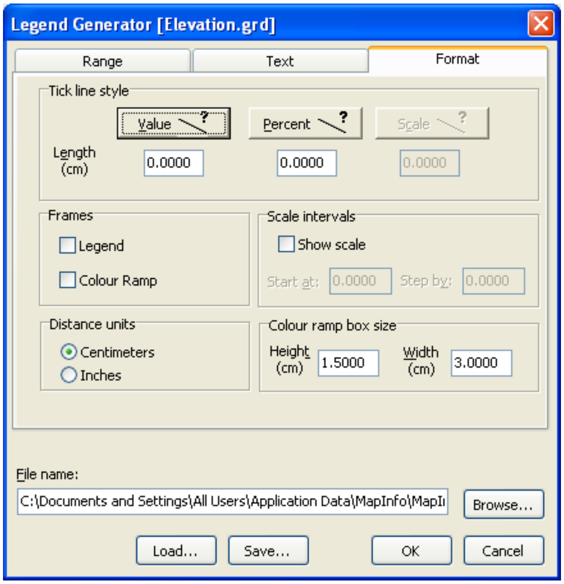
In the **Titles** section, you can enter a **Main Title** and, if desired, a **Subtitle** that will be positioned immediately below the main title. Using the **Other** box, you can also enter another title which is positioned at the bottom of vertical legends and at the lower left base of a horizontal legend bar. You can choose the desired font for each of the three titles by clicking the Text Style button to the right.

The **Label Font** section of the tab enables you to modify the type and size of either the Value label, the Percentile label, or the Scale label by clicking the Text Style button to the right.

The **Label Position** section contains the following options:

- The **Placement** list box enables you to position the labels to the left or right of a vertical legend bar or on the top or bottom of a horizontal legend bar.
- The **Offset** settings control the spacing between the legend bar and the starting point of the tick mark label measured in centimetres or inches.
- The **Rotation** settings control the rotation around the insertion point (upper left) of each text string that comprises a label, where rotation defaults to zero degrees (horizontal) in vertically oriented legend bars and to 90 degrees (vertical) in horizontally oriented legend bars.

The **Format** tab enables you to control the placement and formatting of tick marks, scale intervals, and frames.



The following options are available on the Format tab:

You can use the buttons in the Tick Line Style section of the dialog box to modify the line style and colour of the Value, Percentile, or Scale tick lines. The Length of each set of tick lines may also be set independently. These commands are not available for classified grids.

In the **Frames** section, you can enable one or both of the following options:

The **Legend** check box draws a box around the entire legend.

The **Colour Ramp** check box draws a frame around the entire colour ramp for numeric grids and around each individual colour box for classified grids.

The **Distance Units** section controls the units of distance measurement applied to the non-earth coordinate system of the legend Map window. You can choose either Centimetres or Inches as the unit of measure. The dialog box updates appropriately for settings where distance measurements are required such as tick length or colour box size.

The **Scale Intervals** section controls the placement of a set of tick marks inserted along the entire length of the scale bar at regular intervals. If the Show Scale box is enabled, the Start At setting determines where the scale markings will begin, measured from the start of the colour bar (bottom or left), while the Step By setting controls the interval spacing of the scale markings. The unit of measure is defined by the data value unit. These commands are not available for classified grids.

The **Colour Ramp Box Size** section controls the size of each colour box that makes up the colour legend bar. By default, the numeric grid colour ramp is made up of 64 individually coloured box elements that are set to resemble the current colour settings of the grid. By changing the values in the Height or Width boxes, you can modify the size and proportion of the colour bar.

The **Save** button enables you to save a legend configuration. Legend profiles are saved with a .vml extension.

The **Load** button enables you to load a legend configuration (.vml).

Once all the settings have been made to the colour legend, a file name must be chosen before selecting **OK** to continue. The on-screen grid file display is immediately updated. If a legend option was chosen, this file must also be saved as a .TAB before being displayed in a different Map Window.

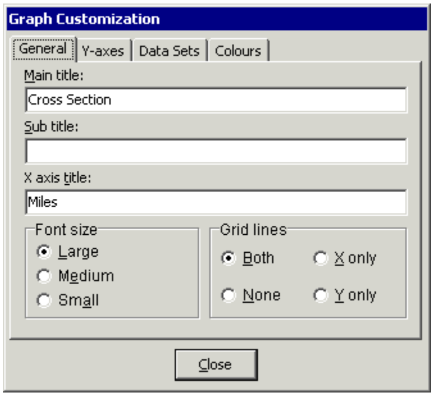
Customizing a Graph

The graphing tool appears as part of four features, namely the grid histogram in the Grid Manager, the Cross Section tool, the Point-to-Point Visibility tool, and the semivariogram in kriging interpolations. Each of these features employ some, if not all, of the available graphing capabilities.

- Position the cursor over a grid histogram, a cross-section graph, or a point-to-point solution graph, right-click and choose the **Customize Graph** command.

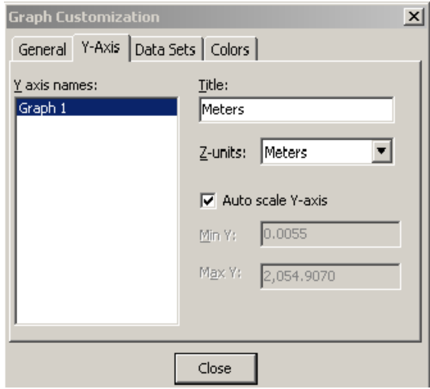
Exploring the Graph Customization Dialog Box

The Graph Customization dialog box contains four tabs that control settings for the graph layout and plot style: the General tab, the Y-axes tab, the Data Sets tab, and the Colours tab.



The **General** tab contains settings that control the text displayed on the graph as well as the graph lines.

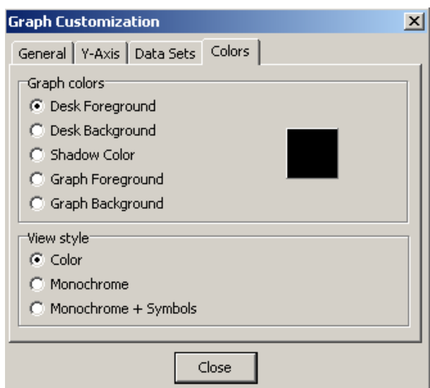
- The **Main Title** and **Sub Title** boxes enable you to enter appropriate titles for the graph window.
- The **X Axis Title** box enables you to assign a title to the x-axis.
- The **Font Size** option buttons enable you to control the font size for all text entities in the Graph window.
- The **Grid Lines** option buttons enable you to choose the placement of the grid lines. You can choose one of the following options: parallel to both the x- and y- axes, on the x-axis only, on the y-axis only, or no grid lines.



The **Y-axes** tab contains settings that control how the labels for each graph in the Graph window are presented. The window to the left of the dialog box displays the number of different graphs that appear in the Graph window. The numbering convention represents the number of graphs that appear in the Graph window starting from the top of the window.

- The **Title** box enables you to change the title of the y-axis for the selected graph. The default is the units of the grid(s) that appear in the graph.
- The **Z-units** list box enables you to convert the units used on the y-axis. This is only available when the z-units of the grids displayed in the graph are a linear measurement.
- The **Auto Scale Y-axis** check box enables you to set the default y-axis labels to the range of data values found in the grid file.

  When you clear this check box, you can enter a custom Min Y and Max Y value for the axis label. If this range is customized so that it is smaller than the range of values used to create the graph, then graph lines may be drawn into adjacent graphs.

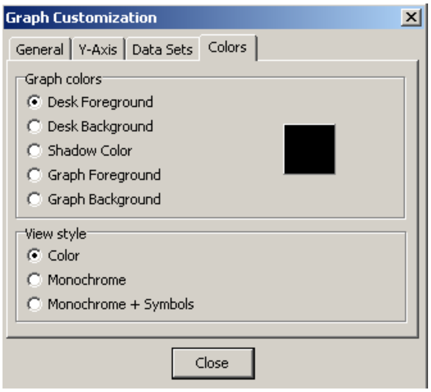


The **Data Sets** tab contains settings that control the style and colour of individual plots in the Graph window. The Data Set Names list displays the names of all the grids that have been plotted in the Graph window. When you choose any grid in the list, the appropriate graph number appears to the right.

- The **Graph Type** list box displays all the plot styles that can be applied to the lines in the graph.
- The **Line Style** list box displays all the line styles for graph types that have line components.

- The **Point Style** list box displays all the point styles for graph types that have a symbol component.

  When you click the coloured button near the bottom of the dialog box, the Color dialog box opens, enabling you to change the plot colour of the chosen grid.



The **Colours** tab contains settings that control the colours of the different components of the Graph window.

- The **Graph Colours** section enables you to control the colour of the Desk Foreground (text entities), Desk Background (the area outside the graph axes), Shadow Colour (the drop shadow for the graph area), Graph Foreground (all grid lines and axis lines), and Graph Background (the area within the graph axes). Colours are modified by choosing an option and clicking the colour button to the right of these options. This section is unavailable if one of the Monochrome view styles is chosen.
- The **View Style** section enables you to choose a Colour plot, a Monochrome plot (black line on white background), or a Monochrome + Symbols plot.

Exporting a Graph

You have three options when exporting a graph:

- The **Text File** option saves all the information used in constructing the graph to a comma-delimited ASCII text file with a .txt extension. The information saved includes the sample number, the distance from the starting point of the line, the x-and-y coordinate of each sample, the overland distance between each sample, and the value at each sample location found on all grids plotted in the graph. The overland distance value is only available when the projection of the grids being graphed is in a Cartesian coordinate system and has not crossed a null data region. When this menu item is chosen, you will be prompted for the name of the text file.
- The **Bitmap & Tab** option saves the Graph window to a 24-bit bitmap image with an associated .tab file. This allows the graph to be opened in MapInfo Professional and placed inside a Layout window. When this menu item is chosen, you will be prompted for the name of the text file. Note that MapInfo Professional versions prior to 5.5 do not support 24-bit images.
- The **Windows MetaFile** option saves the Graph window as a Windows Meta File (*.wmf).

To export a graph:

1. Position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph.
2. Right-click, choose the **Export** command, and choose one of the three Export options: Text File, Bitmap & Tab, or Windows MetaFile.
3. In the Save As dialog box, navigate to the folder where you want to save the file and type the file name.
4. Click the **Save** button.

Printing a Graph

- Position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph, right-click and choose the **Print** command.

The Print command prints the current graph directly to the default printer as set by the computer's operating system.

Maximizing a Graph

- Position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph, right-click and choose the **Maximize** command.

The Maximize command enables you to maximize the graph window to full screen size. You can reduce the graph to the original size by pressing the ESC key or clicking in the upper-left corner of the maximized window.

Zooming In and Out on a Graph

- To zoom in, position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph, then click and drag.
- To zoom out, position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph, right-click and choose the **Undo Zoom** command.

The Undo Zoom command enables you to restore the size of the graph to the original zoom level at any time.

Hiding or Showing the Graph Data

- Position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph, right-click and choose the **Hide Graph Data** or **Show Graph Data** command.

Because two or more graphs may be displayed in the same graph window, the legend indicates the color assigned to each grid file. You can modify the line or symbol color of the graph by double-clicking on the grid file name entry in the legend box. You can then make any changes in the Color dialog box.

Hiding or Showing the Legend

- Position the cursor over a grid histogram, a cross section graph, or a point-to-point solution graph, right-click and choose the **Hide Legend** or **Show Legend** command.

Smoothing a Grid

The Vertical Mapper 3.5 contains a new Smoother feature that reduces the variability in a numeric grid by averaging cell values contained in a specific area. In general there are two categories of filters. The first category includes low pass (averaging) filters used for smoothing the variation of cell values found in a grid. The second category consist of edge detection or high pass filters, these are typically used for enhancing edges in grids and images. The new Vertical Mapper Smoothing feature uses the first category of these filters.

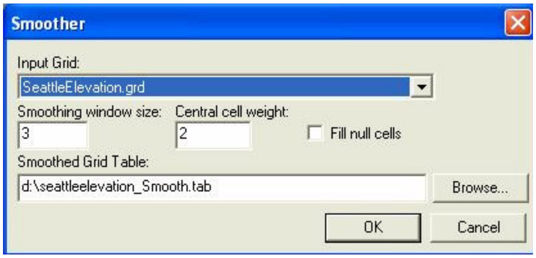
What is Smoothing?

Smoothing is the act of filtering a grid by modifying the values of every cell in a numeric grid to be the average of it's neighboring cells. This averages out the data in the grid to provide a more high-level or relative representation of the data, thus removing any ‘noise’ from contours created from grid data and reducing the size of those contour files.

The basic principal of averaging filters is to pass a window (3x3 or 5x5 cells) through the grid, calculating the average value of all pixels inside the window and assigning this value to the central cell in the window. The size of the window has an impact on the results; a greater degree of smoothing will result when using a larger window size.

Using the Smoother Utility

The Smoother utility only works on numeric grid data. With at least one numeric grid file open, click Smoother from the **Vertical Mapper** menu within MapInfo Professional. The Smoother dialog box displays.



Use the **Input Grid** drop-down list to choose a grid file to smooth from a list of all opened numeric grids.

A default output table file name will be provided in the **Smoothed Grid Table** field based on the name of the selected input grid (“_Smooth” suffix is added). You may change this default file name or click the **Browse** button to select an existing TAB file to overwrite with the new smoothed data. You may want to use the default settings until you are comfortable with the way this utility works.

Use the **Smoothing Window Size** field to control the level of smoothness of the grid. This setting tells the Smoother how big a matrix to look at when computing a grid cell's average neighboring cell value.

The default value is 3, which means a matrix of 3 by 3 cells. This smoothing window size number must be an odd number and range from 3 to 21 (for example, 3, 5, 7, 9, 11, 13, 15, 17, 19 and 21). The 'smoothness' of your contour will depend on how large a value you use; a value of 21 will compute the average of a 21 by 21 matrix, thus yielding a smoother contour due to more grid data used in the average computation for each grid cell. A value of 3 will compute the average of a 3 by 3 matrix, thus yielding a rougher contour due to less grid data used in the average computation for each grid cell.

Use the **Central Cell Weight** field to set the weight of the central cell's value when computing the new averaged cell value. This gives the middle of the contour segment more weight in determining the new path of the smoothed contour segment being defined by the grid.

For example, in the 3 by 3 matrix below,

1 2 1

4 5 3

7 0 9

If the central '5' cell's weight is set as 1, then the new central cell value is calculated as:

1 + 2 + 1 + 4 + (5.1) + 3 + 7 + 0 + 9

--

8 + 1

Where the value 8 in the denominator is the number of cells other than the central cell, and the value 1 both being added to the denominator and being multiplied by the central cell value in the numerator is the central cell weight.

In order to give the central cell value a greater weight, you can enter a weight larger than 1. For example, if you change the central cell weight in the matrix above to 6, the new computed value for that cell is:

1 + 2 + 1 + 4 + (5.6) + 3 + 7 + 0 + 9

--

8 + 6

Check the **Fill Null Cells** check box to have the Smoother compute the average for grid cells that contain no data. You can clear this check box to ignore cells that do not contain data (though this may retain some 'noise' in your output contour data).

Consider the example above with the exception of a null central cell value ('V', meaning 'void' or 'no data'):

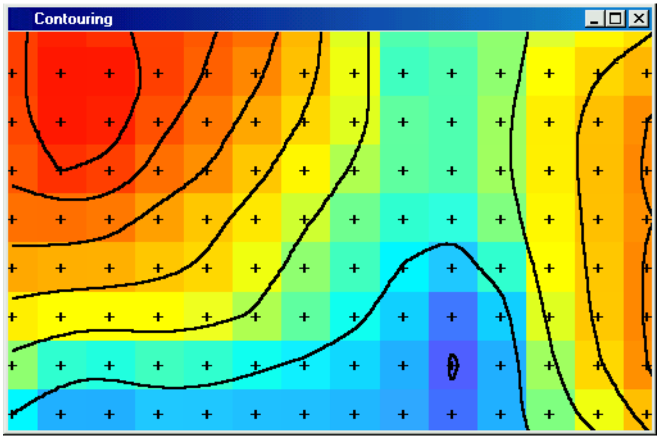
$$\frac{1 + 2 + 1 + 4 + (V.6) + 3 + 7 + 0 + 9}{8 + 6}$$

If you choose to fill null cells, then the central cell (V) will have a new value in the output grid. If you choose not to fill null cells, then the central cell will remain empty (null) in the output grid.

Note The Smoother can significantly reduce the time it takes to complete the contouring process. Use a small matrix size (3x3 or 5x5) and check the **Fill Null Cells** check box prior to contouring your data.

Contouring with Grids

Vertical Mapper provides tools to convert grids to attribute-coded vector files using processes that thread isolines, or contours, through the grid network (as in the next figure). Contour lines are paths of constant values. Vertical Mapper computes contour lines as separate polylines or closed complex regions where holes or islands have been “knocked-out”. This is important in the GIS environment because the contour regions can immediately be used for analysis.



Lines are threaded through grid cells at defined values.

Creating Polyline and Region Contours

A standard contour line map can be generated from within Vertical Mapper using a process that threads polylines through an existing grid file. You can define a number of settings including the range of grid values to be contoured, the contour interval, and the colour and style of individual contour lines. These settings can be saved in a configuration file and applied to other grid files.

You can also generate contours as topologically built MapInfo regions using a process similar to the polyline threader. Contour region intervals are user-defined with the additional option of applying a custom gradient colour ramp to assign incremental colours to the contour regions.

For contour regions, you can define the Greater than or Equal to Lower Value (>=Value) and the Less than Upper Value (<Value) in the Interval List. These values define each contour region based on the Contour settings. Each contour represents the lower value of the interval. For example, the 200 contour region encloses all values greater than or equal to 200 and less than 250.

Defining Contour Polylines or Regions Manually

1. In the Grid Manager, click the **Contour** button. 

2. In the Contour dialog box, choose the appropriate grid file from the **Grid** list and click the **Polylines** button if you want to create polyline contours or the **Regions** button if you want to create region contours.
3. Double-click in any **Value** field in the **Interval List** and type a value.

Defining Contour Polylines or Regions Automatically

1. In the Grid Manager, click the **Contour** button. 

2. In the Contour dialog box, choose the appropriate grid file from the **Grid** list and click the **Polylines** button if you want to create polyline contours or the **Regions** button if you want to create region contours.
3. Click the **Intervals** button.
4. In the Intervals dialog box, do any of the following:
   - In the **Minimum** box, set the minimum value, defined as the lowest grid value that will be examined during the threading process.
   - In the **Maximum** box, set the maximum value, defined as the highest value that will be examined during the threading process.
   - In the **Method** section, choose one of the following options:

     The **Interval** option defines the range of values that each class will encompass. The default setting is calculated by dividing the range between the minimum and maximum values into 10 classes. Use the ***Value*** box to define the interval.

     The **Number** option enables you to specify the number of classes that will be created based upon the Minimum, Maximum and Interval settings. The default is 10 classes. Use the **Value** box to define the number.
5. Click the **OK** button to close the **Intervals** dialog box.
6. To save contour profile settings, click the **Save As** button.

   Profiles are saved as text files with a .pfc extension.

Note The default setting in the Minimum box is the lowest value encountered in the grid file and may need to be changed for the contouring process.

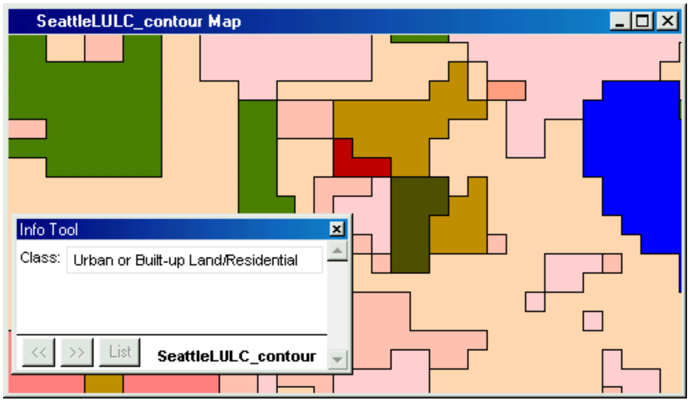
If you want to maintain consistency in contour intervals throughout a series of grid files, save the settings for the contour plot in a profile.

Contouring Classified Grids

Just as region contours are created from numeric grids, they can also be created from classified grids. In this case, a simplified threading process is used to trace line work along the cell boundaries between different classes and convert each unique classified group of cells into a single MapInfo region. The value of each classified grouping is attached as an entry to the region table in a column labeled “Class”.

1. In the Grid Manager, highlight a classified grid file from the list of open grids and click the **Contour** button.
2. In the Save Contours As dialog box, type a name in the **File name** box and click the **Save** button. The contouring function automatically completes and draws the MapInfo region table into a new Map window.

   The new regions are an exact reproduction of the classified grid (.grc) regions but are in MapInfo Professional vector format with an attached column entry.



7

Using Grids for Spatial Analysis

The power of Vertical Mapper lies in the ability it gives you to use grids in complex spatial analysis.

This chapter covers how to:

- use the Grid Calculator
- create grid queries
- use the Point-to-Point Visibility function
- use the Viewshed function
- create slope and aspect grids
- create cross sections
- use the Point, Line, and Region Inspection functions

In this chapter:

Using Grids for Spatial Analysis

Vertical Mapper gives you the ability to create grids based on geographic information and apply this information to complex spatial analysis problems.

For example, annual variations in air temperature can be calculated by subtracting one year's measurements from those of another year. In Vertical Mapper, this arithmetic calculation is performed using the Grid Calculator. Using grids of each year's temperatures, you can subtract the values in one from the values in another to create a new grid with the new values representing the variation. Using the Cross Section tool, you can draw or choose a MapInfo line object that overlies a height grid of the area to construct a vertical profile of elevation along this line.

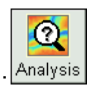
Vertical Mapper includes tools that you can use to solve problems that are virtually impossible to answer using only vector representations of geographic information. These can range from the simple "Can I see that group of buildings from this tower?" to the complex "Given this range of land classifications, this distribution of soil types, these distances from major roads, and these county zoning boundaries, show me the most appropriate sites for a new communications tower."

Note When using functions that query grids such as the grid calculator, grid query, point-to-point visibility, cross section, point inspection, and line inspection, you can choose whether or not values are interpolated within cells in grids by enabling or clearing the **Use Closest Node Value** check box on the z-units tab in the Grid Manager. For more information, see **Working with the Grid Manager on page 89**.

Using the Grid Calculator

The Grid Calculator enables you to create mathematical expressions using an unlimited number of grids. You can modify or filter the gridded data in any one file, or you can generate a new derivative surface from two or more existing grids.

The functions available in the Grid Calculator enable you to perform a number of mathematical operations either on a single grid file (for example, Grid1*sin(25) = Grid2) or on a combination of grids (for example, Grid1/Grid2*Grid3 = Grid4). The new derived grid is the result of applying basic mathematical operations on all of the cell values in an existing numeric grid where each cell is assigned a value satisfying an expression you enter.

1. In the Grid Manager, click the **Analysis** button and choose the **Calculator** command. 

2. In the Calculator dialog box, enter an expression by doing any of the following:
   - Double-click an item in the **Grid/Value** list to enter a grid in the Expression box, or choose a grid in the **Grid/Value** list and click the Grid key on the numeric keypad.
   - Click the required numbers and functions to enter values or operators in the Expression box.
3. If you want to save the expressions to an .exp file, click the **Save** button.
4. If you want to re-use an existing expression, click the **Load** button.
5. In the Grid Math Expression Viewer, choose an expression in the list and click the **Insert** button.

   The expression appears in the Expression box in the Calculator dialog box.
6. In the Calculator dialog box, click the **OK** button.

7. In the Grid Calculator – Save dialog box, do the following:

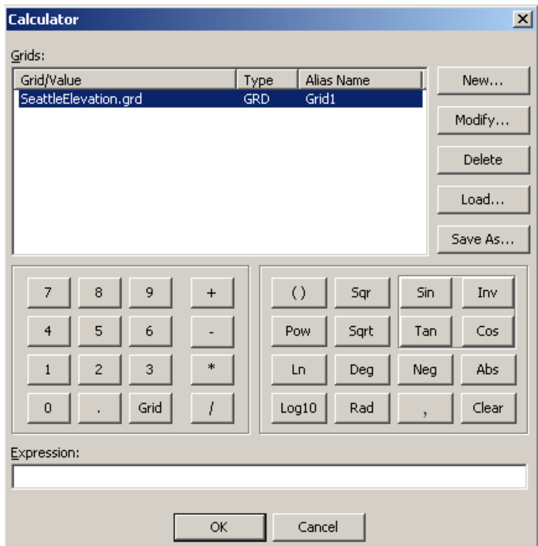
- In the **Save the New Grid As** box, type a file name for the new derived grid built from the math expression.
- In the **Description** box, type a description of the new grid.
- From the **Z-Unit Type** list, choose a z-unit type. If the unit type is user-defined, you can type a custom entry in the **Enter User Defined Typ**e box.

8. Click the **OK** button.

The new grid file appears in a Map window with a default colour palette applied.

Exploring the Calculator Dialog Box

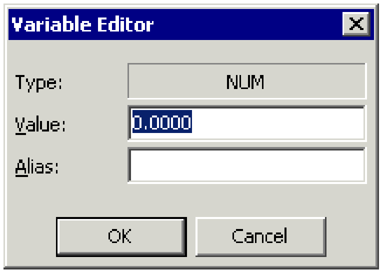
The Calculator dialog box enables you to build mathematical expressions that modify or combine grid files. The number and operator buttons in the Calculator function just like those on a scientific calculator.



The window at the top of the dialog box lists all of the numeric grids currently open in the Grid Manager. An alias name is assigned to each grid or value and is used to identify the item in the **Expression** box.

One major advantage of using an alias instead of the actual grid file name is that the same expression can be applied using different grids.

The **New** button Opens the Variable Editor and enables you to define a new numeric entry and an alias for it. For example, a new value entry could be a numeric constant such as the value of π to 12 decimal places of precision.



- The **Value** box enables you to enter a numeric value.
- The **Type** box displays the type of the variable; this is always NUM for new variables because only numeric variables can be used in expressions with numeric grids.
- The **Alias** box enables you to enter a character string that represents a substitute name for the entry in the Expression box.

The **Modify** button opens the Variable Editor to enable you to edit the Alias of a list entry.

The **Delete** button enables you to delete the chosen entry.

The **Expression** box enables you to build the mathematical expression.

The following table explains the available function buttons, where X refers to a grid or variable name representing a grid.

Button	Usage	Explanation
( )		Adds a left or right bracket
Sqr	Sqr(X)	Square of X
Sin ArcSin	Sin(X) ArcSin(X)	Sine of X Arcsine of X Use the Inv button to toggle between Sin and ArcSin.
Inv		Toggles between Sin, Tan, Cos and ArcSin, ArcTan, and ArcCos
Pow	Power (Base, Exponent)	Raises Base to any power
Sqrt	Sqrt (X)	The square root of X
Tan ArcTan	Tan (X) ArcTan(X)	The tangent of X The arctangent of X Use the Inv button to toggle between Tan and ArcTan.
Cos ArcCos	Cos(X) Arcos(X)	The cosine of X The arcosine of X Use the Inv button to toggle between Cos and ArcCos.
Ln	Ln(X)	The natural logarithm of X
Deg	RadtoDeg(X)	Converts X in radians to degrees
Neg	Neg(X)	Negative of X
Abs	Abs(X)	Absolute value of X
Log10	Log10(X)	Log base 10 of X
Rad	DegToRad(x)	Converts X in degrees to radians
,		Adds a separator
Clear		Clears the expression field

Note The output results for the sine, arcsine, cosine, arcosine, tangent, and arctangent functions are in radians.

Understanding Grid Queries

You can use grid queries to build new grids from existing grids where the new grid values are derived according to whether specific queries imposed on the existing grid files have been met.

For example, you can create an expression that will query a set of grid files and identify all coincident cells that meet these conditions:

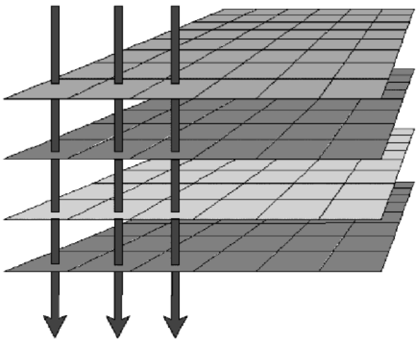
cells in Grid1 are >1049m AND

cells in Grid2 are <=$250,754.50 AND

cells in Grid3 are = “Dense Urban” OR

cells in Grid4 are = “Leeds County”.

Therefore, by examining each grid and “drilling down” through the four grid layers on a cell-by-cell basis, a fifth grid can be built from the overlying and spatially coincident cells that identifies all the cells that meet (or do not meet) all of the query conditions.



The power of grid-based spatial analysis lies in the ability to create a query, using multiple themes of geographic information, that examines on a cell-by-cell basis each layer of data and derives a new grid based on the query conditions.

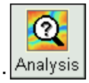
There is no limit to the number of grids that can be used in a Vertical Mapper grid query expression. However, there are restrictions on the specifications of grids that are used in a multiple grid query:

- All grids used in a grid query must be in the same coordinate system.
- All grids must be spatially coincident or at least partially overlap.
- If grids have different cell sizes, the query will be controlled by the grid with the smallest cell size.

The Grid Query tool is comparable to the MapInfo Professional SQL Select command although, in the MapInfo Professional function, the new Query table preserves all of the original data records that meet the SQL conditions. In the Grid Query tool, new cells are created in the derivative grid that are assigned a user-defined value according to whether the cell did or did not meet the conditions of the query expression. This is based on a simple true/false test: Does each cell in the new grid meet or not meet all of the query conditions?

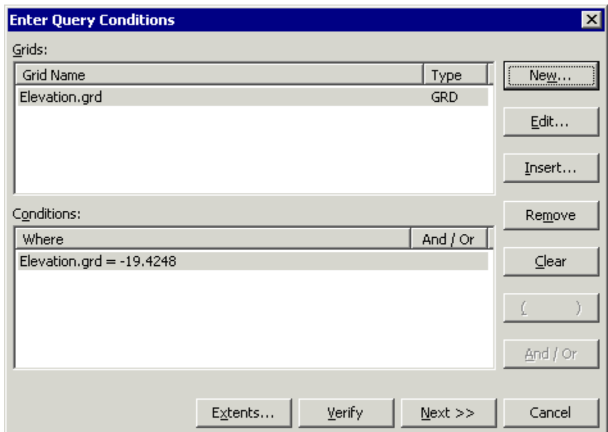
Creating and Editing Conditional Queries

The first step in creating a conditional query involves building a query expression from a selection of open grid files as a series of individual conditional clauses. The second step involves the assignment of values to the new grid, cell by cell, according to whether the query conditions are met (true) or not met (false) for each set of overlying cells.

- In the Grid Manager, click the **Analysis** button and choose the **Grid Query** command. 

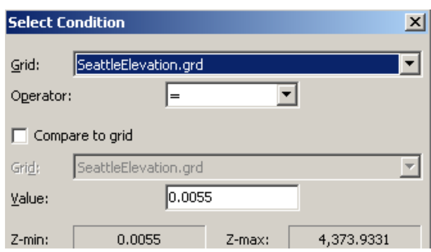
Exploring the Enter Query Conditions Dialog Box

The Enter Query Conditions dialog box enables you to build conditional queries from one or more input grids.



The upper window of the Enter Query Conditions dialog box lists all of the currently open grid files in the Grid Manager. In order to create a new query, you must choose the grid entry.

The **New** button enables you to build a specific conditional query. You can also access this command by highlighting the appropriate grid in the list, right-clicking, and choosing the New Clause command from the shortcut menu. The New button opens the Select Conditions dialog box.



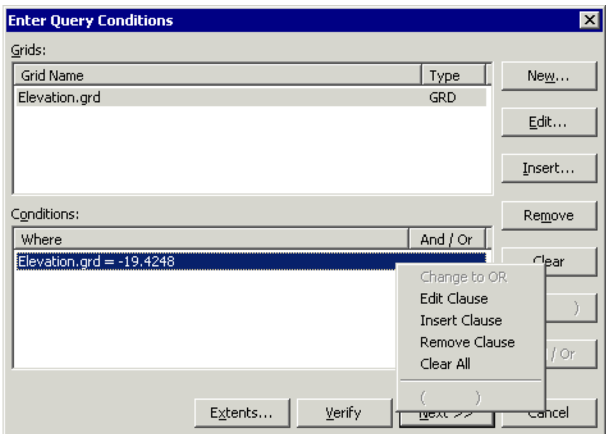
The **Grid** list enables you to select the appropriate grid file to be queried if such selection is not made in the Grid Manager.

The **Operator** list enables you to choose the appropriate operator for the query. The choices available vary depending upon whether the grid is numeric or character-based. In both cases, you can also choose “Any Value” or “Null Value”.

The **Value** box enables you to enter a user-defined value. If the query is being created for a character-based grid, this box becomes a list because there are only a specific number of unique classes in any .grc file.

The **Z-Min:** and **Z-Max**: boxes display the z-value range of any numeric grid file.

You build the query expression in the **Where** box of the Enter Query Conditions dialog box. By default, individual query clauses are separated with an “AND” Boolean operator. The following commands are available to modify a highlighted clause in the Where list and are found both as buttons on the right side of the dialog box and as commands from the shortcut menu for the chosen clause.



The **Edit** button enables you to re-open a Select Condition dialog box for any conditional clause and make appropriate changes.

The **Insert** button enables you to open the Select Condition dialog box and to insert a new clause before the selected one.

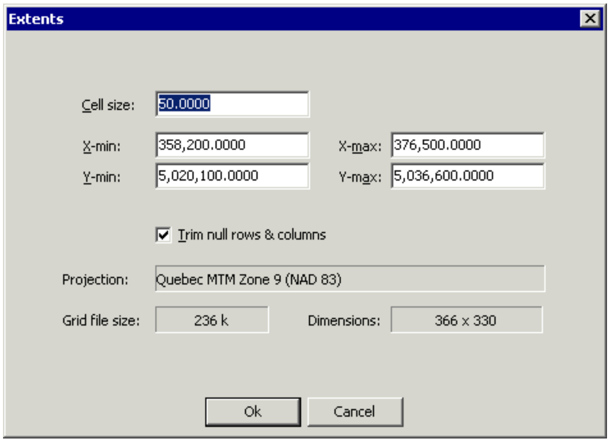
The **Remove** button enables you to delete the selected clause.

The **Clear** button enables you to delete all clauses in the Where list.

The **(   )** button enables you to join and place brackets around two or more clauses and converts them into a single line. This most commonly applies to an expression containing clauses that are joined by both “AND” and “OR” operators where the conditional logic may be unclear. For example, without brackets, the process order is unclear in the expression “GridA = x AND GridY = y OR GridY = z”. The command is only available if two or more clauses are selected in the Where list.

The **And/Or** button enables you to toggle between the AND and OR operators, either of which can join clauses.

The **Extents** button enables you to access the Extents dialog box. This dialog box controls the specifications of the output grid file generated from the Grid Query command.



The **Cell Size** box enables you to set any user-defined value but, as with any grid creation technique, cell size is inversely proportional to file size. This value defaults to the smallest cell size found in all of the open grids.

The **X-min, X-max** and **Y-min, Y-max** boxes enable you to set any user-defined values. They default to the maximum extent covered by the grids used in the query expression.

The **Trim Null Rows & Columns** check box enables you to remove any columns or rows of null value cells from the edges of the grid.

The **Projection** field records the coordinate system of the open grids and is a non-editable setting for the Grid Query command. All grids included in a Grid Query expression must be in the same MapInfo Professional coordinate system as the coordinate system of the output grid.

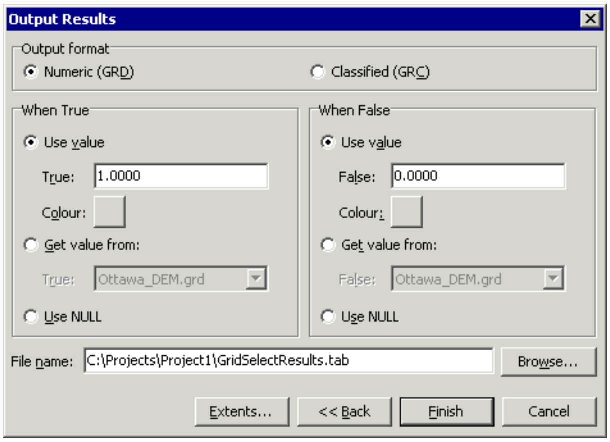
Structuring the Output Results of a Grid Query

The second step of a grid query command requires you to assign values on a cell-by-cell basis to the new grid when the query conditions are met (true) or not met (false) for each set of overlying cells.

Exploring the Output Results Dialog Box

The Output Results dialog box is designed to guide you in making three decisions regarding the form of the output grid file:

- What type of output grid to build: numeric or character-based.
- What value to assign to grid cells that DO meet all of the query conditions (TRUE).
- What value to assign to grid cells that DO NOT meet the query conditions (FALSE).



The **Output Format** section enables you to choose the format of the output grid. You can choose a numeric grid file where each cell in the grid is assigned a numeric value or a classified grid file where each cell in the grid is assigned a character-based value.

The **When True** section contains the following options:

- The **Use Value** option assigns a specific value to each TRUE cell in the new grid that is user-defined and entered into the edit box.
- The **Get Value From** option assigns a value/class to all TRUE cells in the new grid. The value/class given to each TRUE cell in the new grid corresponds to the same cell value/class in the specified grid. This option applies to a specified grid chosen from a list of all currently open files.
- The **Use Null** option assigns null values to all TRUE cells in the new grid.

Note A cell returns a “TRUE” value when it meets all WHERE conditions.

The **When False** section contains the following options:

- The **Use Value** option assigns a specific value to each FALSE cell that is user-defined and entered into the edit box.
- The **Get Value From** option assigns a value/class to all FALSE cells in the new grid. The value/class given to each FALSE cell in the new grid corresponds to the same cell value/class in the specified grid. The **Use Null** option assigns null values to all FALSE cells in the new grid.

Note A cell returns a “FALSE” value when it does not meet all WHERE conditions

The **File name** box enables you to enter a file name for the new grid.

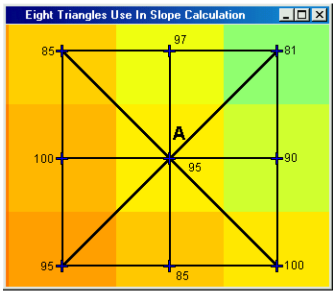
The **Back button** enables you to return to the previous dialog box and make modifications.

The **Finish** button enables you to complete the grid creating process. Once the grid file is created, it opens in a Map window with a default colour palette applied.

Understanding Slope and Aspect

As it applies to grid geometry, slope is a measurement of the “steepness” of a grid cell in three-dimensional space and is therefore most applicable to elevation surfaces. In Vertical Mapper, slope is calculated by averaging the slopes of the eight triangle faces that are formed from the surrounding nodes.

Aspect measures the direction that each grid cell faces in three-dimensional space and is recorded in azimuth degrees relative to either true north or the top of the map. In Vertical Mapper, aspect is calculated by averaging the aspects of the eight triangle faces that are formed from the surrounding nodes.



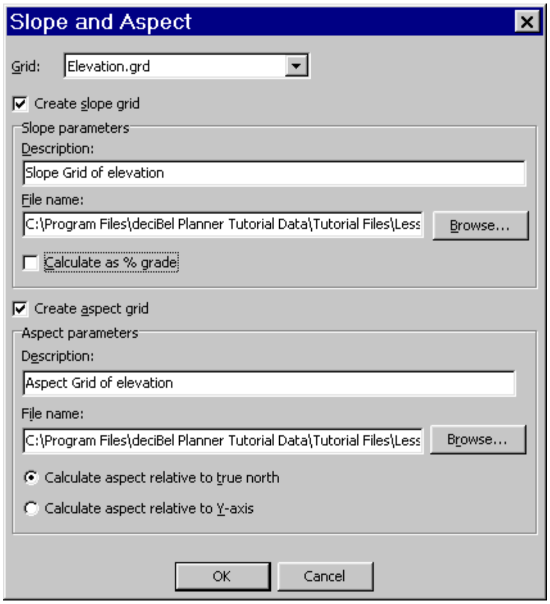
The eight triangles are created to determine the slope at node “A”.

Creating Slope and Aspect Grids

Vertical Mapper enables you to create the slope and aspect for a grid file. The Z-units of the grid must be linear units.

1. In the Grid Manager, click the **Analysis** button and choose the **Create Slope & Aspect** command.
2. In the Slope and Aspect dialog box, choose a grid from the **Grid** list.
3. Enable the **Create Slope Grid** check box to create a slope grid.
4. Enable the **Calculate as % Grade** check box to calculate the slope as a percent grade. When you clear this check box, the slope is calculated in degrees.
5. Enable the **Create Aspect Grid** to create an aspect grid.
6. In the **Aspect Parameters** section, do one of the following:
   - Choose the **Calculate Aspect Relative to True North** option to set north to zero degrees azimuth and allow values to progress in a clockwise direction.
   - Choose the **Calculate Aspect Relative to Y-axis** option to set “Y” at the top of the map.
7. In the **Description** boxes, enter descriptions (maximum 31 characters). The descriptions will be carried as a header in the new grid file.
8. In the **File name** boxes, enter a file name for each grid to be created.
9. Click the **OK** button.

Exploring the Slope and Aspect Dialog Box



You can use this dialog box to create a slope grid, an aspect grid, or both.

The **Grid** list Enables you to choose the list for the analysis.

The **Create Slope Grid** check box Enables you to create a slope grid.

The **Slope Parameters** section This section is available only when you enable the Create Slope Grid check box:

The **Description** box Enables you to enter a description for the slope grid.

The **File Name box** Enables you to enter a new file name for the grid.

The **Calculate as % Grade** check box When this check box is enabled, the slope is calculated as a percentage grade; when it is cleared, the slope is calculated in degrees.

The **Create Aspect Grid** check box Enables you to create an aspect grid.

The **Aspect Parameters** section This section is available only when you enable the Create Aspect Grid check box:

The **Description** box Enables you to enter a description for the aspect grid.

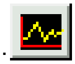
The **File Name box Enables you to enter a new file name for the grid.**

The **Calculate Aspect Relative to True North** option When this option is chosen, north is set to zero degrees azimuth and values progress in a clockwise direction.

The **Calculate Aspect Relative to Y-Axis** option When this option is chosen, “Y” is set at the top of the map.

Creating a Cross Section

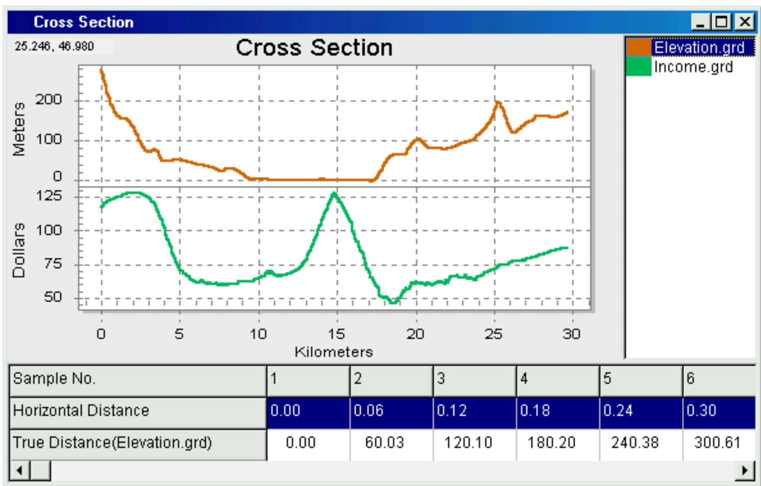
You can create a query of grid values along a line or polyline constructed in a Map window using the Cross Section command. The values are displayed as an x, y line plot in a graph window.

1. On the Vertical Mapper toolbar, click the **Cross Section** button. 
2. Draw the line of the cross section directly on the grid image, then double-click to end the line.

Note You can also create a line of cross section on a selected line or polyline by clicking the Analysis button in the Grid Manager, and then choosing the **Cross Section** command.

Exploring the Cross Section Dialog Box

The Graph window in the Cross Section dialog box contains a number of settings that control display and handling of the graph and legend. The Graph window shows separate plots for all spatially coincident active grids opened in the Grid Manager when grids of different units are encountered. The Graph window has a limitation of six different graphs; however, there is no limit to the number of grids that can be plotted in the same graph.



The distance chart located at the bottom of the Graph window displays three categories of information: the sample number, the horizontal distance, and the true horizontal distance.

Sample No.	1	2	3	4	5	6	7	8
Horizontal Distance	0.00	0.06	0.12	0.18	0.24	0.30	0.36	0.42
True Distance(Elevation.grd)	0.00	60.03	120.10	180.20	240.38	300.61	360.85	421.12

The **Sample No.** identifies each sample taken from the grids along the transect line that was chosen or drawn with the Cross Section tool. The number of samples is defined by the Cross Section Samples setting in the Preferences dialog box.

The **Horizontal Distance** is the "crow fly" distance from the beginning point of the transect line to the indicated sample.

The **True Distance (Elevation.grd)** is the ground or "overland" distance from the beginning point of the transect line to the indicated sample

Note Whenever a cross section line crosses a null area, the true distance is reset to zero.

The Point Inspection Function

The Point Inspection function updates a table of point data with a new column of values taken from one or more geographically coincident grid files. The process inspects the grid file at each point, returns the appropriate grid value, and writes the value to a new column in the point table. In many ways, the procedure is the reverse of creating a grid from a set of points. An example would be adding a column of income data to a point table of dwelling locations using a grid file of average family income.

Using the Point Inspection Function

When you use the Point Inspection function, a new column is created in the point table for each open and active grid that contains the respective grid value corresponding to each point location.

1. Open a MapInfo table of point data and the grid files from which the information is to be extracted.
2. In the Grid Manager, click the **Analysis** button and choose the **Point Inspection** command.
3. From the **Table to Update** list in the Point Inspection dialog box, choose the appropriate table containing the data points to be updated.

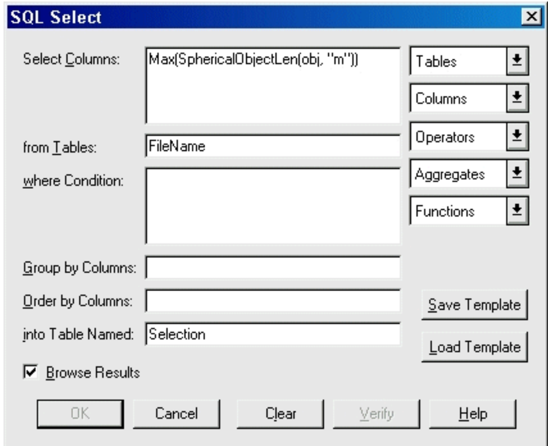
   The **Null Value** edit box displays the value that will be used if the point lies off the grid or over a null cell.
4. Click the **OK** button.

   Note Values from all open and geographically coincident grids will be added as new columns to the point table assuming that all grids are active.

The Line Inspection Function

The Line Inspection function displays several statistical parameters based on the grid values that a specified line overlays, such as the average elevation of a runway. The grid is sampled at several locations along the selected line. Each line is sampled a specified number of times regardless of the line length (the default is 100 samples). Because the number of samples taken influences the final results and each line is sampled the same number of times, statistics calculated for very long lines may be less accurate than those calculated for shorter lines.

The way to get around this is to determine how long the longest line is and then base the sample number upon that. As shown in the next figure, a simple MapInfo Professional SQL Query can be performed to do this, and the number of samples can be modified in the Preferences dialog box.



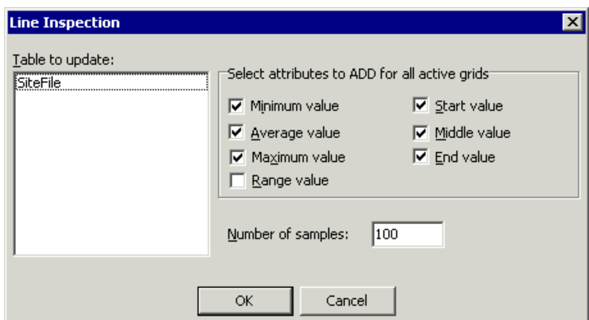
This figure shows a MapInfo Professional SQL Query.

Using the Line Inspection Function

The Line Inspection function updates a table of polyline data with values taken from a grid file. The process inspects the samples between the beginning and end of the line, calculates the selected number of statistical parameters, and writes the values to new columns in the data table.

1. Open a MapInfo table of line data and the grid files from which the information is to be extracted.
2. In the Grid Manager, click the **Analysis** button and choose the **Line Inspection** command

.



3. From the **Table to Update** list, choose the appropriate table containing the data lines to be updated.
4. In the **Select Attributes to ADD for All Active Grids** section, choose all of the appropriate statistical parameters (values) and the number of samples that will be calculated for each line and written into the data table. The default number of samples is 100.
5. Enable any or all of the **Start Value, Middle Value**, and **End Value** check boxes to add columns to the table with the values at the start, middle, and end of the line.
6. Click the **OK** button.

Each statistical parameter calculated for each open and active grid is written into a separate column of the region table and given a default column name consisting of both the parameter name and the grid file name.

Note The number of samples taken along the line is determined by the Cross Section: No of Samples parameter in the Preferences settings. The default is 100. To change this value, choose the Vertical Mapper, Preferences command and enter a new number in the No of Samples box. For more information about Preferences, see **Setting your Preferences on page 204**.

The Region Inspection Function

It is important to understand the manner in which Vertical Mapper calculates area using grids. Every grid file is constructed of cells of equal area, as determined from the unique coordinate system specifications to which the grid has been projected. However, a grid in any coordinate system is only truly accurate, in terms of real spherical (ground) distance, along the line of standard parallels. Therefore, as you move farther away from the standard parallel, the actual ground area of a grid cell becomes less accurate (usually it is larger than its real spherical area). While in most cases the error will not be significant, if area is important, you have two options to minimize this error:

- Ensure that the grid is always projected to an appropriate coordinate system that is relevant to the geographic region of interest.
- Using the Vertical Mapper Contour function, convert the grid file covering the area of interest to a MapInfo region and then query the region to obtain MapInfo Professional's spherical-based area calculation.

Using the Region Inspection Function

The Region Inspection function updates a MapInfo table of regions with new columns of values taken from one or more geographically coincident grid files. The process inspects the grid file underlying each region, returns a selected number of statistical parameters calculated from the range of grid values lying within each region, and writes the value to a new column in the region table.

1. Open a MapInfo table of point data and the grid files from which the information will be extracted.
2. In the Grid Manager, click the **Analysis** button and choose the **Region Inspection** command 

.
3. In the Region Inspection dialog box, choose the appropriate region table to update.
4. In the **Select Attributes to ADD for All Active Grids** section, choose all of the statistical parameters that will be calculated for each region and written into the region table.
5. Click the **OK** button.

Each statistical parameter calculated for each open and active grid is written into a separate column of the region table and given a default column name consisting of both the parameter name and the grid file name.

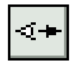
The Point-to-Point Visibility Function

Intervisibility and viewpoint analysis uses elevation grid files to determine visual exposure relationships within a map area. Intervisibility is defined as the ability to see in a direct line of sight from one position on the earth's surface to another, considering the intervening terrain.

The Point-to-Point Visibility function enables you to specify a line across an elevation grid file (i.e., digital elevation model) for the calculation of intervisibility. The calculation returns an answer that is both simple in response: "the two points ARE/ARE NOT intervisible", and more complex: "this is the height that one of the two points would have to be raised to become visible", or "this is the height that one of the two points would have to be lowered to become visible".

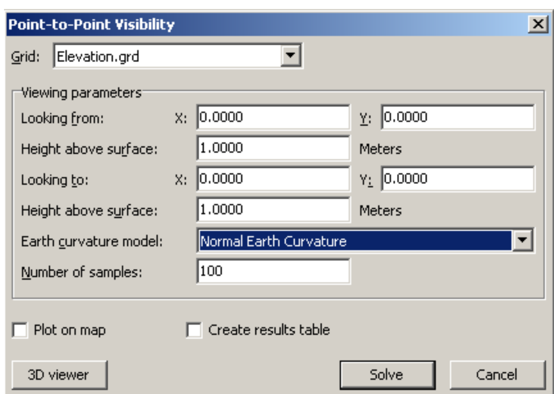
Using the Point-to-Point Visibility Function

The Point-to-Point Visibility function enables you either to select a line-of-sight path from an existing line object in a Map window or to draw the line-of-sight path directly in the Map window. You can use this function only on a numeric grid that has a z-unit type of feet or metres.

1. On the Vertical Mapper toolbar, click the **Point-to-Point Visibility** button .
2. With the left mouse button held down, draw the line-of-sight path in the Map window of the open elevation grid file, where the line direction corresponds to the direction of sight, that is, the "looking from" position is the start of the line.
3. In the **Viewing Parameters** section of the Point-to-Point Visibility dialog box, choose all of the parameters that will be calculated for each region and written into the region table.
4. Click the **Solve** button.

Exploring the Point-to-Point Visibility Dialog Box

The Point-to-Point Visibility function is appropriate for use on a grid file which has a unit of elevation (feet or metres) as its z-value. If the Viewshed command encounters a grid file where the unit type is not recognized as being a unit of elevation measurement, a warning appears, and you will be unable to proceed with the command.



The **Grid** list enables you to select the appropriate open grid file for analysis if such selection is not made in the Grid Manager.

The **Viewing Parameters** section enables you to control the intervisibility calculation for each endpoint of the line.

You can enter x- and y-coordinates as well as an offset height above the surface at both the Looking From and the Looking To viewpoint positions. This section enables you to add a value to the From point that represents, for example, the height of a transmission tower. Similarly, a height value may be added to the To site that represents, for example, the height of a receiver.

If you are trying to measure line of sight from a transmitting antenna to a receiving car antenna, you may want to add a value of 120 metres to the From point to represent the height of the transmission tower, and add 1.5 metres to the To point to represent the approximate the height of the car antenna above the surface of the ground.

The **Earth Curvature Model** list enables you to choose an earth curvature model. Earth curvature must be taken into account for most line of sight calculations. You can select one of the following models:

Normal Earth Curvature calculated using an oblate spheroid model.

4/3 Earth Curvature allows radio frequency engineers to model the slight curvature of a radio wave as it travels over the earth's surface due to diffraction effects.

No Earth Curvature ignores earth curvature

The **Number of Samples** box enables you to control the number of points along the line of sight from which the intervisibility calculation is made and, therefore, the number of points that will be plotted in the line of sight graph.

A value of 100 is considered appropriate for most calculations; however, the greater this value, the more detail is added to the graphed profile.

The **Create Results Table** check box enables you to save all of the graphed information to a MapInfo table, which can be viewed in a Browser window.

VMPTP1 Browser

Distance	Viewable_Elevation	Surface_Elevation	Relative_Viewable_Elevation	Relative_Elevation
0	50.1815	50.1815	50.1815	50.1815
232.248	50.7653	50.7653	50.761	50.761
464.496	50.8176	50.8176	50.8006	50.8006
696.744	49.8057	49.8057	49.7675	49.7675
928.992	47.6608	47.6608	47.5931	47.5931
1,161.24	45.2166	45.2166	45.1107	45.1107

The resulting table contains five columns with the following information.

- The **Distance** value is the distance measured from the viewing location to the sample location. The sample location is the location along the viewing line where the grid was analysed. By default this line is sampled 100 times. However, it can be modified in the Point-to-Point dialog box.
- The **Viewable Elevation** value is the elevation at the sampled location that can be seen from the viewing point. All values are based on the "no earth curvature" model, and they are not displayed in the graph window.

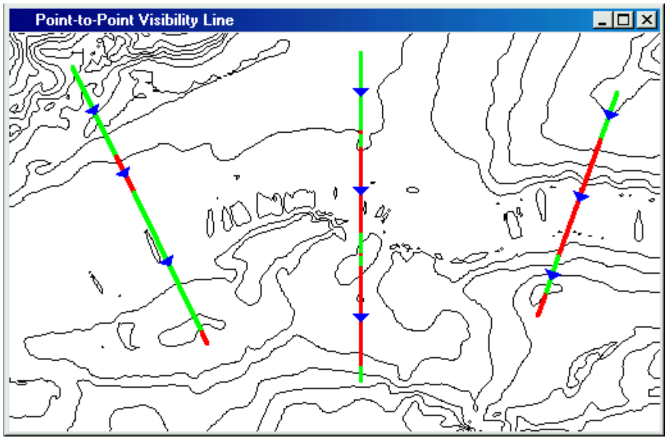
- The **Surface Elevation** value is the elevation value found at the sampled location. All values are based on the “no earth curvature” model, and they are not displayed in the graph window.
- The **Relative Viewable Elevation** value is the elevation at the sampled location that can be seen from the viewing point. All values are based on the selected earth curvature model and are represented by the green line in the graph window.
- The **Relative Elevation** value is the elevation value found at the sampled location. All values are based on the selected earth curvature model and are represented by the red line in the graph window.

If the No Earth Curvature model is chosen, then the Viewable Elevation and the Relative Viewable Elevation columns will be the same. Also, the Surface Elevation and the Relative Elevation will be the same.

The **Plot on Map** check box enables you to view a line plot in the Map window after clicking the Solve button.

The line plot traces the extent of the line of sight and indicates, using colour, the intervals between the endpoints that are visible (green) and obstructed (red) relative to the direction of sight. Line plots are saved as individual MapInfo .tab files using default file names (subdirVMLineX.tab). As subsequent lines are chosen and solved, new files are saved with incremental numbers in the file name.

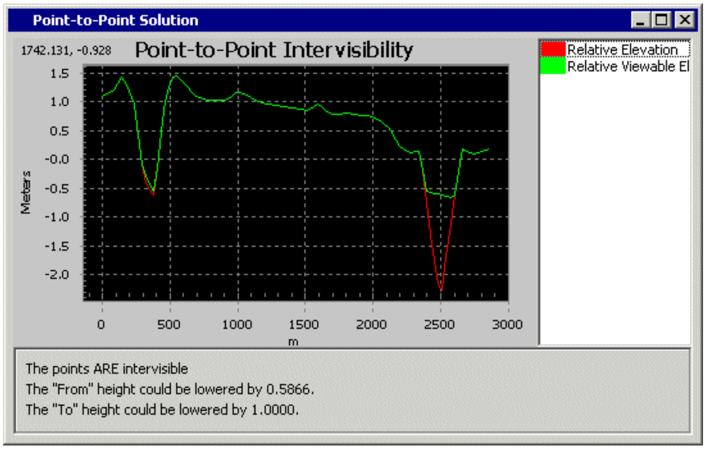
The **Solve** button enables you to initiate the point-to-point calculation.



The point-to-point visibility line

Exploring the Point-to-Point Solution Dialog Box

The Point-to-Point Solution dialog box is a graphical representation of the line-of-sight calculation. Two lines of cross section are constructed in the graph window along the line of visibility.



The relative elevation indicated by the red line represents the surface topography along the line of sight, while the relative viewable elevation (green) represents the line of visibility. Where the two lines are coincident, that section of topography is visible from the “From” point.

Immediately below the graph is a written description of the relationship between the “From” and “To” points of the line of sight path. If the two end points of the line are intervisible, the dialog box will report how the “From” and “To” heights could be adjusted and still remain visible. If the line end points are not intervisible, the dialog box will report the corrections necessary to produce intervisibility.

The **Solve** button enables you to initiate a new calculation every time a different line of sight path is constructed using the Point-to-Point tool.

The Viewshed Function

Viewshed is defined as a delineation process identifying all locations on a grid that are visually connected (visible in a direct line-of-sight) to a single observation point.

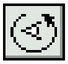
The Viewshed function computes visibility between one or more observation points (the viewpoints) and each of the cells in an elevation grid file (the destination cells) in one of two ways: as a simple visible/invisible answer for each of the destination cells; or as a computed value representing the height that each destination cell should be raised or lowered to make it just visible from the viewpoint.

In other words, if a destination cell is not visible from the viewpoint, then a negative value is returned specifying the height below the line of sight. To become just visible this height has to be added to the destination cell. If the destination cell is visible, then a positive value is returned specifying the height above the line of sight. The viewpoint can be lowered by this height and remain just visible.

The Viewshed function is designed to work with the Viewpoint Pick tool found on the Vertical Mapper toolbar. You can use the tool to identify the view point from which intervisibility for an entire elevation grid file is calculated or you can use it to select an existing point present in the Map window.

Using the Viewshed Function

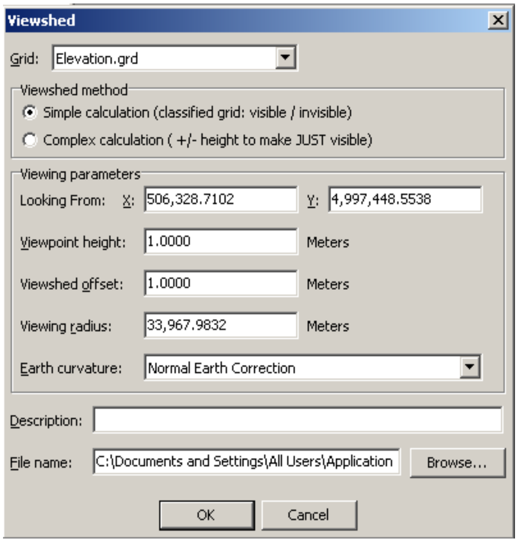
The Viewshed function is appropriate only for use on a grid file that has a unit of elevation (feet or metres) as its z-value.

1. On the Vertical Mapper toolbar, click the **Viewpoint Pick** button. 
2. Using the left mouse button, choose a point in the Map window of the open elevation grid file that represents the point of origin for the viewshed calculation.
3. In the Viewshed dialog box, enter the required parameters in the appropriate boxes and click the **OK** button.

   Note You can also access the Viewshed tool from the Grid Manager. Click the **Analysis** button, and choose the **Viewshed Analysis** command. You can use this method to do a multi-point viewshed analysis. For more information on multi-point viewshed analysis, see **Performing Multipoint Viewshed Analysis on page 134**.

Exploring the Viewshed Dialog Box

The Viewshed dialog box enables you to set a number of viewshed parameters that control the calculation.



The **Grid** list enables you to select the appropriate grid file for analysis if such selection is not made in the Grid Manager.

The **Viewshed Parameters** section enables you to calculate the visibility values assigned to each cell in the new Viewshed grid in two ways:

- The **Simple Calculation** option enables you to create a classified grid file and assigns either the category “Visible” or “Invisible” to each cell depending upon whether it is visible or invisible from the viewpoint.

  If multiple viewpoints are selected, each grid cell of the new classified grid is assigned a category “NumVisible_n”, where “n” is the number of viewpoints visible from that cell; the values will range from zero to the total number of viewpoints used in the calculation.
- The **Complex Calculation** option returns a value measured in grid z-units.

  The value represents either the height the cell should be raised to make it just visible from the viewpoint (a negative value because it lies below the sight line), or the height that the viewing points should be lowered in order to become just visible from the grid cell (a positive value because it lies above the sight line).

The **Looking From** box enables you to set the point on the map from which all intervisibility calculations are made (the viewpoint). The position is determined by entering known coordinates or by using the Viewpoint Pick tool.

The **View Point Height** box enables you to set the value for viewpoint height if, for example, the viewpoint represents an observation or a transmission tower of known height. The height is automatically displayed in metres.

The **Viewshed Offset** box enables you to add a height to account for the size of an object that is being viewed.

For example, if you want to measure a line of sight from a transmitting antenna to a receiving car antenna, you may want to add a value of 1.5 metres at every grid location to better approximate the true height of the car antenna above the surface of the ground. The height is automatically displayed in metres.

The **Viewing Radius** box enables you to control the maximum radius around the viewpoint for which Viewshed is calculated.

The **Earth Curvature Model** list enables you to choose an earth curvature model. Earth curvature must be taken into account for most line of sight calculations. You can select one of the following models:

Normal Earth Curvature calculated using an oblate spheroid model.

4/3 Earth Curvature allows radio frequency engineers to model the slight curvature of a radio wave as it travels over the earth’s surface due to diffraction effects.

No Earth Curvature ignores earth curvature.

Note If the grid is in latitude/longitude, you must choose a linear unit from the list. For all grids defined in latitude/longitude, you must have a defined earth curvature.

Performing Multipoint Viewshed Analysis

You can perform viewshed analysis using several points.

1. In the map window, using any of the MapInfo Professional select tools, select point objects representing the locations you want to perform the analysis on.
2. In the Grid Manager, click the **Analysis** button, and choose the **Viewshed Analysis** command.
3. In the Viewshed dialog box, enter the settings you want to use for the analysis and click the **OK** button.

8

Using the Grid Tools

The grid tools are a series of utilities that work with grid files to increase the flexibility and workability of grids in the MapInfo Professional desktop mapping environment.

In this chapter:

Using the Grid Tools

Vertical Mapper provides a selection of grid tools that enable you to export grids to other mapping formats, reproject grids from one coordinate system into any other MapInfo Professional-supported system, reclassify grid values to predefined values or classes, splice adjacent or coincident grids into a seamless map coverage, trim a grid to fit into a custom map boundary, and change the resolution of a grid to reduce the file size or to make it more comparable to adjacent or overlying grids.

These tools are designed to directly support you in working with the entire range of analytical functions available in Vertical Mapper. In addition, these functions make grids more usable as maps for sharing information visually and for transferring geographic data between GIS environments.

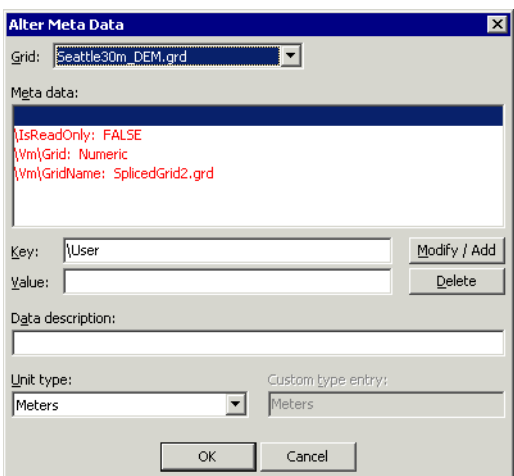
Using the Alter Meta Data Tool

The Alter Meta Data tool provides you with the ability to append information to the existing grid meta data. This can be particularly useful when it is necessary to retain other information about a grid file, such as the name of the project or the name of the person who created it. The Alter Meta Data tool also enables you to modify the header of a grid with a new data description and/or data unit. This saves time because it allows modifications to be made to any grid without having to regenerate it. This tool is particularly useful for adding data descriptions to grids that originate from sources other than Vertical Mapper and for altering a grid so that it can be used with the Point-to-Point Visibility and Viewshed functions. Both of these tools can be used only on grids in which the data unit is a distance measurement, for example, metres or feet.

1. In the Grid Manager, highlight the grid to be modified.
2. Click the **Tools** button and choose the **Alter Meta Data** command.

Exploring the Alter Meta Data Dialog Box

Using the Alter Meta Data dialog box, you can add information to the existing grid meta data.



The **Grid** list enables you to select the appropriate open grid file to process if it is not chosen in the Grid Manager.

The top portion of the dialog box contains a list displaying the current meta data for the grid. The text colour, either black or red, indicates whether entries are editable (black) or not editable (red). You cannot edit entries containing Vertical Mapper or MapInfo Professional keys. The blank entry at the top of the list must be highlighted when you add new information. All new entries will appear in alphabetical order in the list.

The **Key** box enables you to identify any new information that is added to the meta data. The default Key is “User”; however, this can be replaced by any text string.

The **Value** box enables you to enter new information. The text string can be a maximum of 239 characters.

The **Modify/Add** button enables you to add the current information or update the selected information.

The **Data Description** box enables you to enter a new grid description up to 31 characters. This description will appear in the z-units tab of the Grid Manager.

The **Unit Type** list enables you to specify the unit of the values contained in the grid.

If you choose the User Defined option, you must type a value in the Custom Type Entry box. The User Defined data type is always used for grids with z-values measured in units other than distance units. This feature does not perform any mathematical calculation on the grid when you change the unit type. As a result, only the information contained inside the grid file is changed.

Exporting Grids

The Export Grid tool is designed to support the transfer of numeric and classified grid data created in Vertical Mapper to external applications. In most cases, the grid node x- and y- location coordinates and the z-value attribute attached to each node are extracted from the grid file and written to the new format. Currently, Vertical Mapper supports ten export file types.

- ASCII Point Export (.txt)—x, y, z space delimited text file*
- 3D DXF Point File—x, y, z AutoCAD points file*
- 3D DXF Mesh—AutoCAD mesh file*
- 3D DXF Mesh (Microstation)—Microstation-compliant DXF mesh format*
- VRML (.wrl)—a simple Virtual Reality Markup Language “world” of the grid file*
- MapInfo Point Table (.tab)—point table with z-value attribute column
- Windows Bitmap (.bmp & .tab)—image built according to the display settings of the grid. The file name is appended with “_bmp”.
- ASCII Grid Export (.txt)—text file
- TIFF (.tif)—Tagged Image File Format image
- USGS DEM**

*supports only numeric grid export

**USGS DEM format specifications require Grid Z-Units to be in Meters or Feet

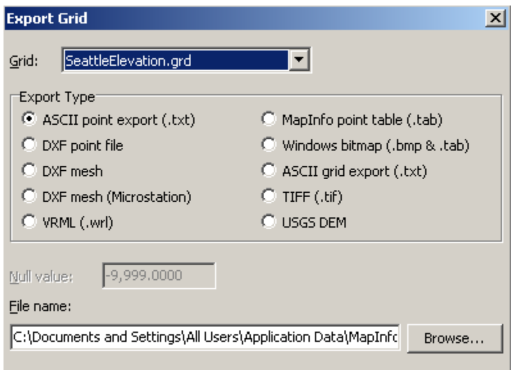
Using the Export Grid Tool

1. In the Grid Manager, click the **Tools** button and choose the **Export** command. 

2. In the Export Grid dialog box, from the **Grid** list, choose the grid you want to export.
3. In the **Export Type** section, choose the export type.
4. If you are exporting to either of the DXF mesh formats or the VRML format, specify a null value in the **Null Value** box. This differentiates a “real” zero value from a null value in the export file.
5. In the **File name** box, type a new file name or accept the default.
6. Click the **OK** button.

Exploring the Export Grid Dialog Box

The Export Grid dialog box provides the interface to the Export tool.



The **Grid** list enables you to select the grid file you want to export.

The **Export** section enables you to specify the type of file you want to export to.

The **Null Value** box enables you to specify a null value for the two DXF mesh formats and the VRML format. It is not applicable to other formats.

The **File Name** box enables you to specify a file name for the exported grid.

Using the Classified Grid Filter

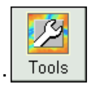
Using the Classified Grid Filter, you can reclass small isolated areas of cells. This has the effect of smoothing the grid. The classes are processed individually, beginning with the class containing the smallest number of grid cells.

1. In the Grid Manager, click the **Tools** button and choose the **Classified Grid Filter** command.

2. In the Classified Grid Filter dialog box, from the **Grid** list, choose the grid to be filtered.
3. In the **Number of Stranded Cells** box, type the number of cells that denotes an area to be reclassified.
4. Disable the **Use Largest Neighbour** check box and type the ratio of the number of cells in the adjacent class to the perimeter of the area to be reclassified in the **Neighbour Uniformity Threshold** box.
5. In the **File name** box, type a new file name or choose the default.
6. Click the **OK** button.

Displaying Grid Legends

The Grid Legends tool enables you to display the legend of a selected grid and quickly access the colour tool so that you can modify grid colours.

1. In the Grid Manager, click the **Tools** button and choose the **Grid Legends** command. 

2. In the Grid Legends window, from the list at the bottom of the window, choose the grid for which you want to display a legend.
3. Click the ellipsis (***...***) button to the right of the list to modify the legend.

   For numeric grids, you modify the legend using the Grid Colour Tool, and for classified grids, you modify legends using the Dictionary Editor.

Reclassing Numeric Grids

You can reclass the values contained in a grid in order to make the data more suitable for analysis. When you reclass a grid, you generate a new grid. You can save reclass profiles for reuse with other grids using the Save button.

1. In the Grid Manager, click the **Tools** button and choose the **Reclass** command.. 

2. In the Reclass dialog box, choose the grid you want to reclass from the **Grid** list..
3. Choose the format of the output grid by clicking the **Numeric** or the **Classified** button in the top right corner of the dialog box.

4. Do one of the following:
- Click the **Insert** button to insert a row in the Reclass table.
- Click the **Delete** button to delete the selected row in the Reclass table.
- Click the **Interval** button to specify the minimum and maximum for the range or interval.

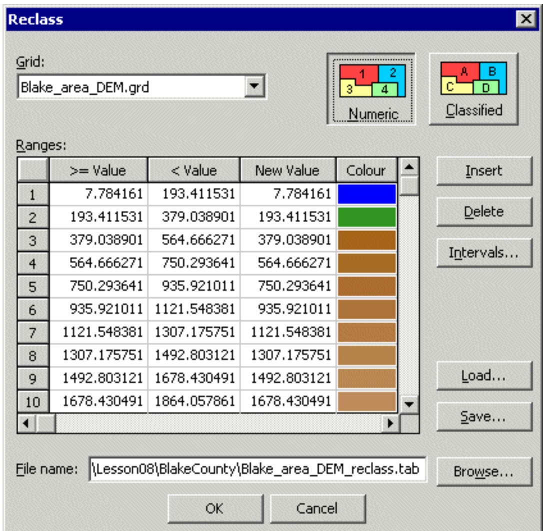
5. Click the **Save** button to save the reclass profile to a .pfr file.

6. In the **File name** box, type a new file name or accept the default.

7. Click the **OK** button.

Exploring the Numeric Grid Reclass Dialog Box

The Numeric Grid Reclass tool enables you to reclassify the values contained in a numeric grid so that it will be more suitable for analysis.



The **Numeric** and **Classified** buttons enable you to specify the grid type for the resulting grid. For numeric grids, the classes have numeric values beginning with lowest value in the range. For classified grids, the classes have descriptions beginning with Class 1. You can change values by double-clicking in a field.

The **Intervals** button opens the Interval dialog box, which enables you to define the following values:

- The **Minimum box** enables you to define the lowest value in the grid that will be used in the reclassification. Any cells with a value less than the Minimum setting will receive the null value in the resulting grid. This setting defaults to the lowest value in the grid.
- The **Maximum box** enables you to define the highest value in the grid that will be used in the reclassification. Any cells with a value greater than the Maximum setting will receive the null value in the resulting grid. This setting defaults to the highest value in the grid.

- The **Method** section enables you to choose which method to use to create the range. There are two options:

  Interval in the Value box, type the range of values that each class will encompass. The default setting is based on dividing the range between the minimum and maximum values into 10 classes

  Number in the Value box, type the number of classes that will be created based upon the Minimum, Maximum and Interval settings. The default is 10 classes.

The **Load** and **Save** buttons enable you to load existing settings and to save the current settings so that they can be used in subsequent grid reclassifications. The save process creates an ASCII text file with a .pfr extension containing the range, the value, and the colour of each class.

The **Ranges** table displays the range of values being classified, class name/new value, and the class colour. Class ranges are created with equal spread and include the lower value and up to but not including the higher value. You can specify the new class name/value by editing the adjacent class name cell.

Reclassing Classified Grids

You can reclass the values contained in a classified grid to make the data more suitable for your analysis. When you reclass a grid, you generate a new grid. You can save a reclass profile for reuse with other grids using the Save button.

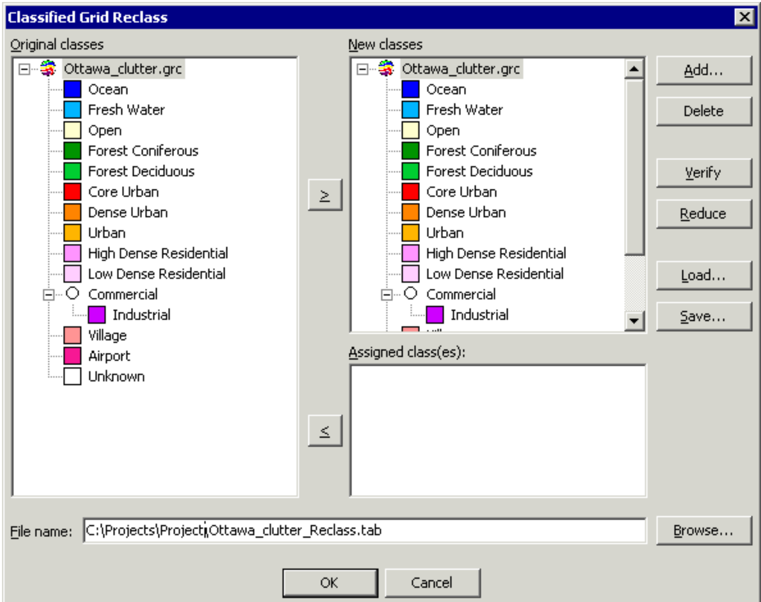
1. In the Grid Manager, highlight the grid you want to reclassify. 

2. Click the **Tools** button and choose the **Reclass** command.
3. In the Classified Grid Reclass dialog box, do any of the following:
   - To assign a class, select it in the Original Classes window and drag it to the New Classes window. Groups cannot be moved over.
   - To add a new class or group, click the **Add** button.
   - To remove the selected class, click the **Delete** button.
   - To move an existing class out of a group, drag the class to the top of the **New Class** list and drop it on top of the file name.
   - To check that no classes are unassigned, click the **Verify** button.
   - To remove unassigned classes from the **New Classes** list, click the **Reduce** button.
4. To save the reclass profile to a .pfr file, click the **Save** button.
5. In the **File name** box, type a new file name or accept the default.
6. Click the **OK** button.

   Note All classes must be reassigned before you can generate the new grid.

Exploring the Classified Grid Reclass Dialog Box

The Classified Grid Reclass dialog box enables you to select and reassign classes or groups from the Original Classes list to the New Classes list. You can do this either by clicking and dragging original classes to new ones or by clicking the Add button. A small triangle marker appears beside the class name in the Original Classes list if it has been successfully reassigned to a new class. When a class is highlighted in the New Classes list, all of the classes assigned to it will be listed in the Assigned Class(es) list. It also makes it possible for you to verify whether any particular original classes or groups are reassigned to new classes or groups.



The **Original Classes** list lists all the classes and groups contained in the original classified grid.

The **New Classes list** lists the new classes that the new classified grid will contain.

The **Assigned Class(es)** section lists classes that you have assigned.

The **Add** button enables you to create new classes or groups and add them to the new grid to be reclassified.

The **Delete** button enables you to remove any highlighted class or group from the new grid to be reclassified.

The **Verify** button enables you to verify that no class or group is left unassigned. If any classes from the original grid are left unassigned, a dialog box opens listing those classes.

The **Reduce** button enables you to remove unassigned classes from the New Classes list.

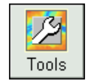
The **File name** box enables you to enter a new file name for the reclassified grid.

The **Load** button enables you to load an existing structure profile.

The **Save** button enables you to create an ASCII text file containing the current classification structure displayed in the New Classes list.

Using the Reproject Tool

The Reproject tool enables you to transform a grid from its existing coordinate system to another MapInfo Professional supported system, which results in the interpolation of new values for each new grid cell. The process involves a complex coordinate transformation that makes use of quadratic surfaces to transform x, y locations in the original grid to the new coordinate system using four cell blocks. The procedure has been engineered to be accurate to within five percent of the new grid cell size at any location across the new grid extent. For example, if the new cell size is 100 metres, the cell location will be accurate to within ±5 metres. The process uses a variation of the bilinear interpolator to recalculate new cell values at each transformed grid cell.

1. In the Grid Manager, click the **Tools** button and choose the **Reproject** command 

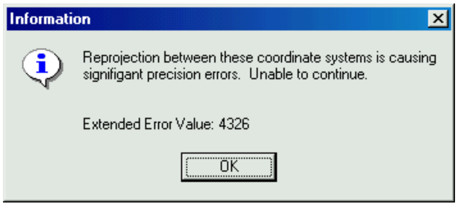
.
2. In the Reproject Grid dialog box, choose a grid from the **Grid** list and click the **Projection** button.
3. In the Choose Projection dialog box, choose a projection category and category member and click the **OK** button.
4. Click the **OK** button to close the Information dialog box containing a suggested range of cell size values.

   This range of values represents the minimum and maximum cell sizes to which any of the original grid cells will be transformed over the area of the new reprojected grid. If you select a new cell size for the reprojected grid that is less than half as small as the minimum recommended value, then the process will return a warning message stating that “This cell size is not recommended.” This warning is meant to ensure that grid cells in the new reprojected file approximate as closely as possible the size of the original cells. The same warning message will appear if you select a new cell size for the reprojected grid that is greater than twice the maximum recommended value.
5. In the **File name** field of the Reproject Grid dialog box, type a new file name or accept the default.
6. Click the **OK** button.

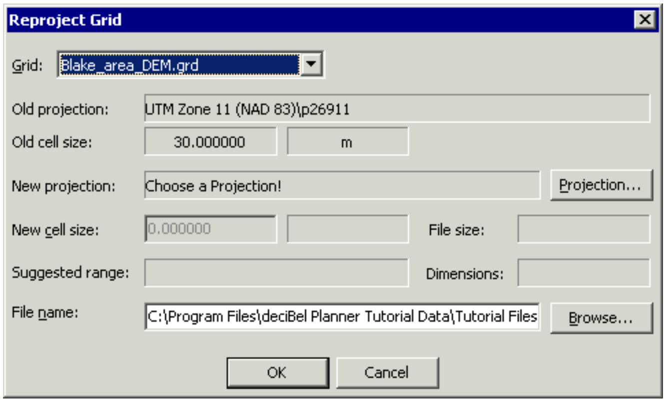
The Reproject tool enables you to perform a preliminary transformation test using the chosen projection.

The test calculation involves projecting the bottom left and upper right points of the original grid into a new projection and then back again. The difference in the position between the original and transformed sets of locations is measured and compared to the original. Most coordinate transformations return error values less than 0.0000001%. However, not all transformations are considered “appropriate” and may be characterized by greater error.

For example, you may want to reproject a map covering a portion of North America in Universal Transverse Mercator (cylindrical) projection to a Lambert Azimuthal (conic) projection centred on the South Pole. This would return a transformation error approaching 10 percent. The reproject algorithm will return a message if the transformation test returns an error >0.02 percent and will discontinue the calculation.



Exploring the Reproject Grid Dialog Box



The **Grid** list box enables you to choose the name of the grid to reproject.

The **Old Projection** box displays the original projection of the grid.

The **Old Cell Size** box displays the original cell size of the grid.

The **New Projection** box displays the new projection or the message “Choose a Projection!” if the new projection has not yet been chosen.

The **Projection** button opens the Choose Projection dialog box, enabling you to choose the new projection.

The **New Cell Size** box enables you to specify a new cell size.

The **Suggested Range** box displays the suggested range of cell sizes appropriate for the projection.

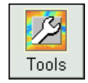
The **File Size** and **Dimension** boxes display the file size and dimensions of the grid in the new projection.

The **File Name** box enables you to specify the file name.

Resizing a Grid

You can adjust the resolution of a grid. For example, you might need to reduce the size and complexity of grid files in preparation for contour threading. While a very high resolution grid file is suitable for modeling and high quality output, a lower resolution version of the same file is adequate for generating contour lines or regions. When the cell size of an existing grid is increased, a new value must be interpolated for every cell in the new grid. You can visualize the original smaller cell grid as a network of nodes overlain by a new network consisting of more widely spaced nodes. The value at the new grid node is calculated using a bilinear interpolation (rectangular interpolation) of the four nearest nodes of the underlying, more closely-spaced grid.

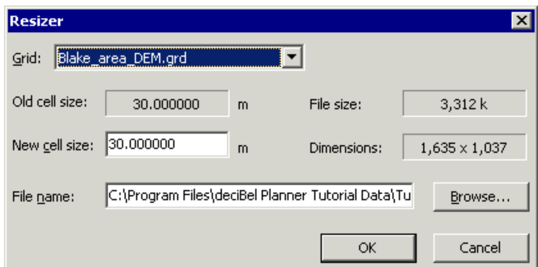
The file size increases when you resize a grid to a higher resolution. While a high-resolution grid is not more accurate, reducing the grid cell size will generate a more highly smoothed surface and shows a continuous gradient colour fill over the grid area.

1. In the Grid Manager, click the **Tools** button and choose the **Resizer** command 

.
2. From the **Grid** list, choose the grid you want to reclass.
3. In the **Resize Parameters** section, enter a new cell size in the **New Cell Size** box.
4. In the **File name** box, type a new file name or accept the default.
5. Click the **OK** button.

Exploring the Resizer Dialog Box

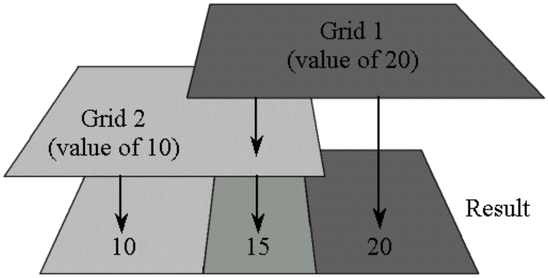
The Resizer dialog box enables you to set the parameters for a new grid. The **New Cell Size** box enables you to specify a new cell size.



Splicing Grids Together

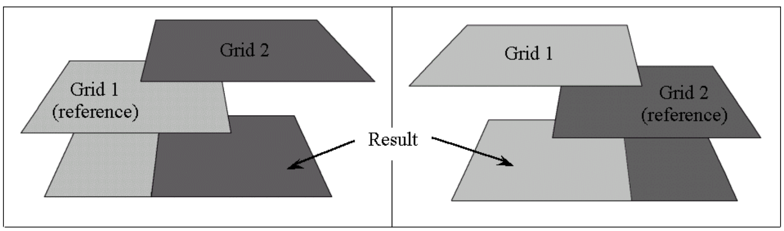
The Grid Splicer tool combines one or more grids into a single grid file. There are two distinct ways in which grids can be spliced together: merging and stamping. The difference lies in how each of these procedures handles the values of grid cells that overlap other grid cells.

In merging, two or more grids are combined, and where they overlap a mathematical calculation is performed to determine the new grid value. Only numeric grids can be merged. You can choose from five calculation methods: Average, Minimum, Maximum, Average (Min, Max) (average of the minimum and maximum values), or Median, where Average is the default. Therefore, if two grids are merged using the Average calculation, and where the first grid has cells with the value of 20 and the second has cells with the value of 10, then any overlapping areas will have cells with the value of 15 in the resulting grid (next figure). Merging is often used when several grid tiles cover a study area that needs to be combined into a single grid, or when you want to create a grid that has the highest/lowest value of all the input grids. Null values contained in any of the grids being processed are ignored.



In the areas where Grid 1 (value of 10) and Grid 2 (value of 20) overlap, the resulting grid will have a value of 15 when the Average overlap calculation is chosen.

In stamping, two or more grids are combined, and where they overlap, the values of one of the input grids will overwrite the values of the other. You can stamp both numeric and classified grids, but grids can be stamped only with a grid of similar type. It is not possible to stamp a classified grid with a numeric or vice versa. Stamping is usually performed when you need to update areas of a grid, for example, updating a land use grid with the area of a new subdivision or an elevation grid with building heights. Null values contained in any of the grids being processed are ignored.



Depending on which grid is the reference grid and the order in which the input grids are listed, different results will be obtained. On the left, Grid 2 is being stamped on Grid 1. Therefore the cell values of Grid 2 will override the values of Grid 1 in the resulting grid. On the right, Grid 1 is being stamped on Grid 2. Therefore the cell values of Grid 1 will override the values of Grid 2 in the resulting grid.

You can update specific areas of an existing grid. This is best done by creating polygonal regions around the areas to be updated, i.e., the new subdivision or building footprint. Attach the appropriate information to these regions, for example, "urban" for the subdivision and the building heights to the footprint. Use the Region to Grid command to convert these regions to a grid. Be sure that the projection and the grid units are the same. Use the Grid Splicer command to stamp these "region grids" on the grid to be updated.

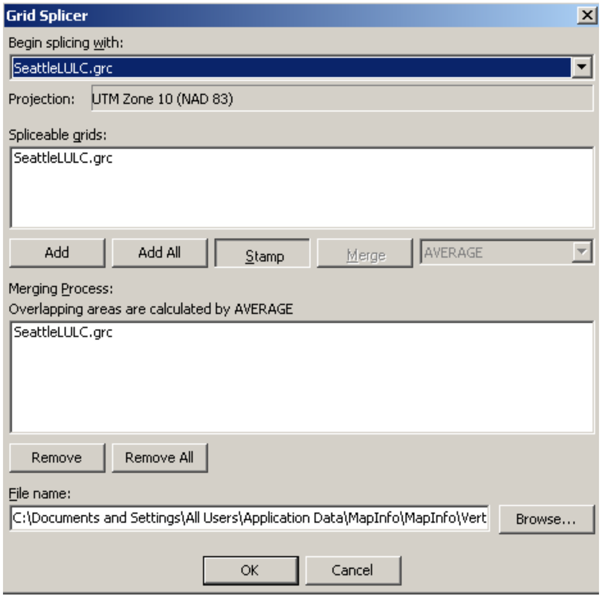
1. In the Grid Manager, click the **Tools** button and choose the **Splicer** command .

2. In the Grid Splicer dialog box, choose a grid from the **Begin Splicing With**: list.
3. From the **Spliceable Grids** list, choose a second grid and click the **Add** button.

4. Choose the type of splicing by doing one of the following:
   - Click the **Stamp** button to combine two or more grids. In the overlapping areas, the cell values of one input grid will overwrite the values of the other grids.
   - Click the **Merge** button to combine two or more grids and choose a calculation type from the associated list. In the overlapping areas, a mathematical calculation is performed to determine the new grid value.
5. In the **File name** box, type a new file name or accept the default.
6. Click the **OK** button.

Exploring the Grid Splicer Dialog Box

The Grid Splicer dialog box enables you to manage all of the grid files used in the splicing process.



The **Begin Splicing With** list enables you to specify which grid will be used as the reference grid for the splicing operation.

The **Spliceable Grids** list displays all grid files open in the Grid Manager that can be successfully spliced with the reference grid.

This list is restricted to those grids that are the same grid type (numeric or classified), the same projection, and the same z-units as the reference grid. There are no restrictions placed upon the cell size of the input grids. The resulting grid will always have the same cell size as the reference grid.

The **Add** button enters the grid highlighted in the **Spliceable Grids** list in the **Merge Process** list. If you have a number of grids to enter, you can select them using the CTRL key and then use the **Add All** button to add all the selected grids to the **Spliceable Grids** list.

The **Stamp** and **Merge** buttons enable you to specify what type of splicing will be performed. Only one of these processes can be performed at a time.

The **Stamp** button enables you to stamp two or more grids.

You cannot choose which mathematical calculation will be performed on the overlapping regions. Grid values in the overlapping regions will be determined by the order in which grids are listed in the Merge Process list.

The **Merge** button enables you to merge two or more grids.

You must specify the mathematical operation that will be performed on the grid cells contained in the overlapping regions of the input grids. Choose one of the following options from the list: Average, Minimum, Maximum, Median, or Average (Min, Max) (average of the minimum and maximum values). These options are only available for splicing numeric grids and are applied to all grids in the splicing process. It is not possible to use different operations on different grids in the same splicing operation.

The **Merge Process** list displays all the grid files that will be used in the splicing operation, and the reference grid will always be listed first.

A message will appear above the window reminding you what splicing process is chosen and displaying a brief description of how the grids will be processed. The order in which the grids are entered is only important when a Stamp is specified. The reference grid is stamped on the results grid and the results grid is stamped on the third grid. The reference grid is always the top grid in the Merge Process list. This process is repeated for each of the grids listed. The order in which the grids are listed can be altered by choosing a grid and then dragging it to a different position in the list.

The **Remove** and **Remove All** buttons enable you to remove the highlighted grid or all the grids in the Merge Process list. The reference grid will always remain in the window.

The **File name** box enables you to enter a unique file name for the spliced grid.

Trimming a Grid

The outer margin of any grid file can be trimmed using a predefined polygon created in a Map window. This tool is particularly useful for trimming grid files to a standard map neatline. Only one grid file can be trimmed at a time.

1. Create the trim region using the MapInfo Professional **Polygon** tool on an editable layer in a Map window or choose the trim region from an open map file. If the region is not already a polygon, choose it and use the **Convert to Regions** command from the **Objects** menu in MapInfo Professional.
2. Choose the region that represents the trimming edge.
3. In the Grid Manager, click the **Tools** button and choose the **Trimmer** command. 

.
4. In the Save Trimmed Grid As dialog box, type a new file name for the trimmed grid in the **File name** box.

Once the .grd file is created, it appears in a Map window with a default colour palette applied. You can change the colour range assigned to the grid file. For more information, see **Using Colour in Grids on page 96**.

Note You can also trim a grid by choosing the Trim button on the Vertical Mapper toolbar and then choosing the region to trim with.

To trim grid cells lying outside a region, that is, knocking a “hole” out of a grid coverage, create a complex MapInfo region. Use the Objects command from MapInfo Professional in which the outer boundary lies outside the grid area and the inner boundary corresponds to the “knock-out”, then apply the trimmer command to trim the grid cells lying within the knock-out.

9

Data Analysis

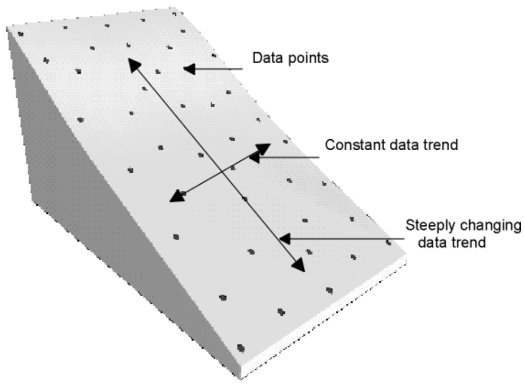
Surface Analysis in Vertical Mapper provides data analysis tools that make your analysis tasks easier.

In this chapter:

Semivariogram Analysis

Grid creation is a valuable way to visualize how data, represented by points, changes through space. The change in value between known locations can easily be shown. However, one thing a grid does not show very well is how values change in any given direction and how strong that directional trend is. This can be particularly valuable to know in some fields. For example, in mineral exploration applications this information is important when trace elements have been transported to their present location from the source by stream, landslide, glacier, or wind. The transportation of the mineral often produces linear orientation in the landscape, and if it can be properly modeled, the source can be found. Direction can also be important in understanding the geographic relationships of demographic data such as average family income and population density.

A directional trend is the tendency for data points with similar values to be arranged in a linear fashion or in a particular direction. Often there is more than one directional trend; however, generally one trend dominates. The most popular technique for examining directional trends is variance analysis.



Example of data that does not vary crosswise but varies greatly along the y-axis of the data.

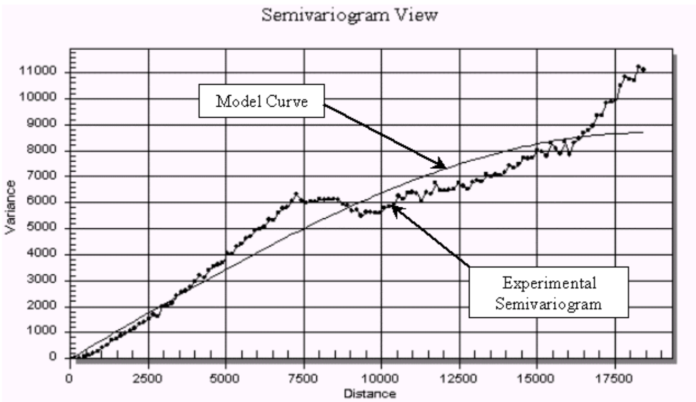
Semivariance expresses the degree of relationship between points on a surface. The semivariance is simply half the variance of the differences between all possible points spaced a constant distance apart.

The semivariance at a distance d = 0 will be zero because there are no differences between points that are compared to themselves. However, as points are compared to increasingly distant points, the semivariance increases. At some distance, called the range, the semivariance will become approximately equal to the variance of the whole surface itself. This is the greatest distance over which the value at a point on the surface is related to the value at another point. The range defines the maximum neighbourhood over which control points should be selected to estimate a grid node, taking advantage of the statistical correlation among the observations.

The calculation of semivariance between sample pairs is performed at different distances until all possible distance combinations have been analyzed. The initial distance used is called the lag distance, which is increased by the same amount for each pass through the data. For example, if the lag distance is 10 metres, the first pass calculates the variance of all sample pairs that are 10 metres apart. The second pass calculates the variance of all sample pairs 20 metres apart, the third at 30 metres and so on until the last two points that are the farthest apart have been examined.

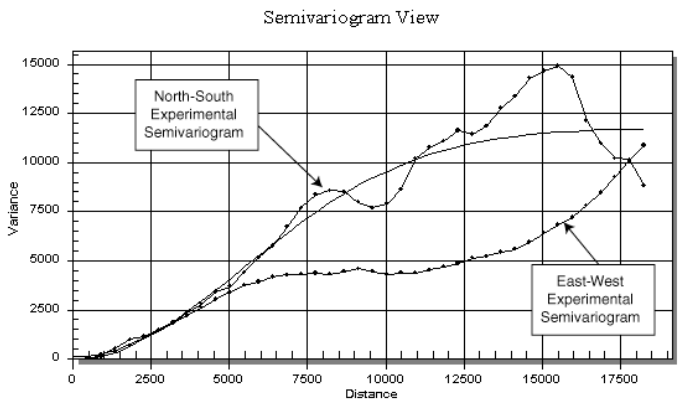
Put simply, every point is compared to every other point to determine which points are approximately the first lag distance apart. When points this distance apart are found, the variance between their values and their geographical orientation is determined. Once the first lag distance has been analyzed the process is repeated using the second lag distance and then the third, and so on until all distance possibilities are exhausted. When the variance analysis is completed, the information is displayed in a semivariogram.

A semivariogram is a graph that plots the semivariance between points on the y-axis and distance at which the semivariance was calculated on the x-axis. An example of a semivariogram is shown in the next figure. The undulating line on the graph is the plot of calculated semivariances, plotted on the y-axis, and their corresponding lag distances on the x-axis. This plot is given the term experimental semivariogram. The jagged nature of the experimental semivariogram makes it unsuitable for use in applications such as calculating kriging weights, so a smooth mathematical function (model) must be fit to the variogram. The model is shown as the smooth line on the graph.



An example of an omni-directional semivariogram.

Although the strength of semivariogram analysis is its ability to account for directional trends of the data, it is possible to analyze variance with respect to distance only and disregard how points are geographically oriented. The above experimental semivariogram is an example of this, called an omni-directional experimental semivariogram. If the geographic orientation is important, then a directional semivariogram should be calculated such as the one shown in the next figure.



An example of a directional semivariogram. Notice the two experimental semivariograms, one representing points oriented north and south of each other, and the other representing points oriented east and west of each other.

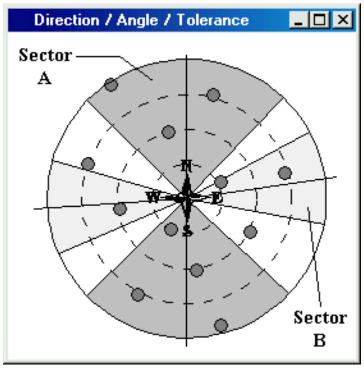
When two or more directions are analyzed, an experimental semivariogram will be generated for each direction. In the previous figure, two directions are being investigated and therefore two experimental semivariograms are plotted.

Semivariogram experimentation can uncover fundamental information about the data, such as whether the data varies in more than one direction. In more technical terms, the semivariogram experimentation can reveal whether the data is isotropic (the data varies the same in all directions) or anisotropic (the data varies differently in different directions) as demonstrated in in the previous figure.

When investigating these directional trends, you will have to modify parameters such as the directions in which the variances will be calculated. These parameters are discussed in the following section.

Modifying Directional Parameters

Up to this point the directional calculation of variance has been discussed as being north-south and east-west. In reality, data will not have directional variations that are described in these exact directions. Therefore, you will need to create a model that “looks” in the direction in which the data is varying. This is done by modifying the number of different directions analyzed, the angle in which they are oriented, and the degree of tolerance that will be afforded to each direction.



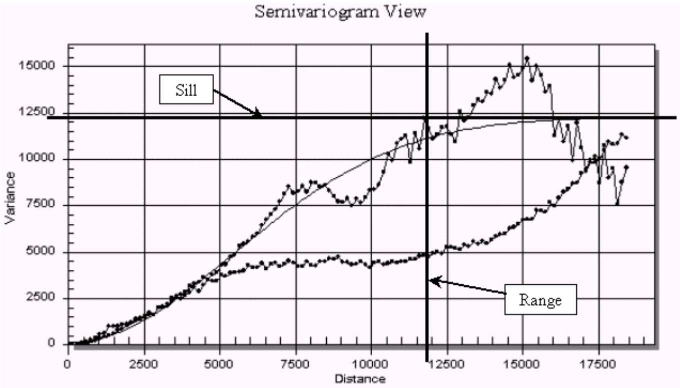
A two-directional semivariance search.

In the previous figure, two directions are analyzed, represented by the dark and light grey pie shapes. It is important to note that although the diagram shows four pies, variance analysis is always performed in opposing directions. When more than one direction is set, the angle to which these sectors will be oriented must be specified. In the above diagram, the angles are zero degrees (Sector A) and 80 degrees (Sector B). It is unlikely that data pairs will be found at exactly 0 degrees or 80 degrees, thus you will need to define an interval around these exact values for which points will be considered. This interval is known as the tolerance. In the above diagram, the zero degree direction has a tolerance of 45 degrees and the 80 degree direction has a tolerance of 20 degrees.

Tuning the Model

After you have generated the experimental semivariogram, you can calculate a model curve that closely fits the semivariogram.

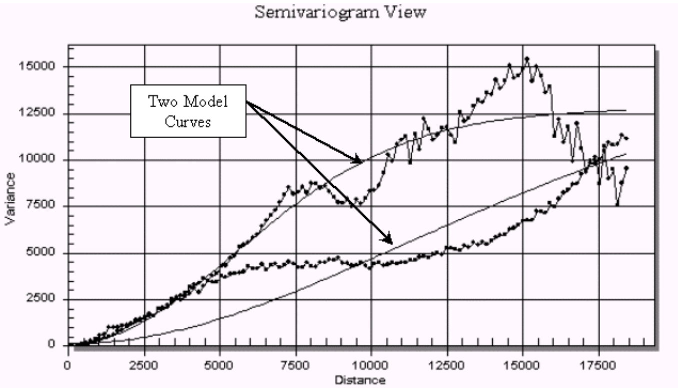
The semivariogram models included with Vertical Mapper are Spherical, Exponential, Gaussian, Power, Hole Effect, Quadratic, and RQuadratic (Rational Quadratic). When one or more of these models is applied to the different directional semivariograms, the model curve can be adjusted to better represent the variance in the data. After any of these models are applied, they can be further modified by changing the sill and range values. The range is the greatest distance over which the value at a point on the surface is related to the value at another point. Variance between points that are farther apart than the range does not increase appreciably. Therefore, the semivariance curve flattens out to a sill. Not all data sets exhibit this behaviour. In the next figure, the sill value is at variance of 12 200, and the range value occurs at a distance of 12 000.



The sill is a variance value that the model curve ideally approaches but does not cross. The range is the distance value at which the variogram model determines where the sill begins.

Anisotropic Modeling

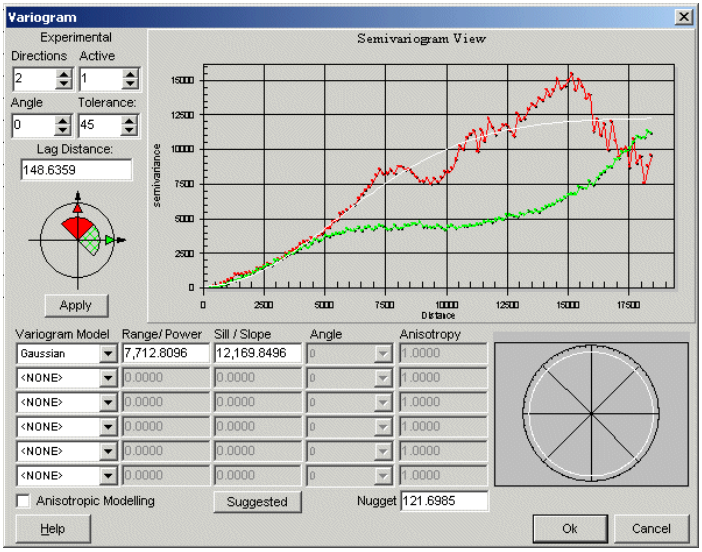
It is quite natural for the behaviour of a data set to vary differently in one direction as compared to another. For example, a steeply sloping hill will typically vary in two directions. The first is up and down the hill where it varies sharply from the top to bottom, and the second is across the hill where it varies more gradually. When this occurs in a data set, it is called anisotropy. When performing anistropic modeling, you are essentially using sample data points that will most accurately reflect the behaviour of the surface. This is achieved by creating additional models for each direction analyzed. When interpolating points are oriented in a north-south direction, kriging weights can be influenced to use the parameters of one model while the points oriented in an east-west direction will be weighted using a different model.



A two-directional, two-model semivariogram.

Exploring the Variogram Dialog Box

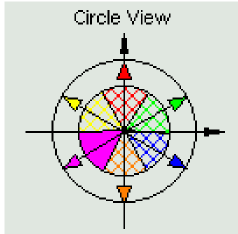
This dialog box is available from the Vertical Mapper, Data Analysis, Semi-variogram Analysis command and from the Variogram Builder button in the Kriging Interpolation dialog box. The top portion of the dialog box displays the experimental semivariogram of the data along with settings that control the directional calculations. The bottom portion of the dialog box contains settings that tune the model. By default, a “best-fit” model is automatically calculated, and you can further modify it manually.



The **Experimental** section contains the following options:

- The **Directions** box enables you to specify the number of directions that will be analyzed. The default value is two with a minimum of one and a maximum of six.
- The **Active** box enables you to specify which direction is currently being modified. The active direction is shown by a solid fill colour in the circle view diagram.
- The **Angle** box enables you to specify the angle in degrees with respect to true north in which the active direction is facing.
- The **Tolerance** box enables you to set the interval in degrees on either side of the angle setting within which points will be considered.
- The **Lag Distance** box enables you to specify the distance at which sample pairs will be analyzed for variance. This value increases until every point in the data has been examined. The default value is the mean distance between points for the aggregated data.

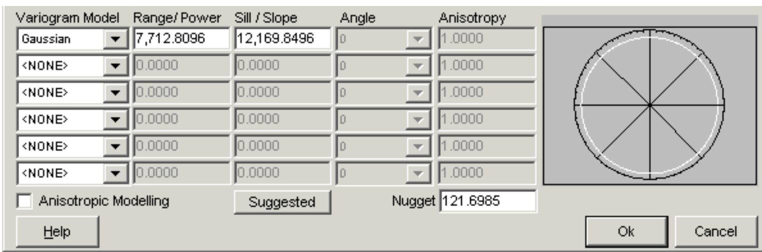
The **Circle View** diagram represents each of the specified directions and their respective tolerances. The solid coloured region represents the active direction for which any changes to the angle or tolerance settings will be applied.



The **Apply** button enables you to recalculate and refresh the semivariogram using new settings.

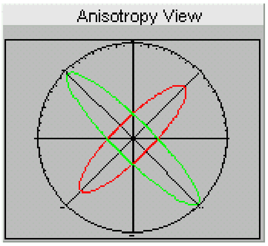
The graph in the dialog box is a semivariogram of the data and plots variance between sample pairs on the y-axis and the lag distances for the calculated variances on the x-axis. The experimental semivariograms that appear in the graph are updated only when you click the Apply button. The model curves will automatically update when changes that affect the models have been made to the dialog box. You can modify the visual display of the graph. You can maximize it to full screen by applying a zoom, or you can undo the zoom. You can also switch from colour to black and white. You can access these options by right-clicking while the cursor is positioned over the graph.

The bottom portion of the dialog box enables you to modify the model curve so that it better fits the experimental semivariogram. This is done by applying one or more variogram models to the model curve.



- **Variogram Model** reflects the model that will be applied to the model curve. Seven variogram models are available: Spherical, Exponential, Gaussian, Power, Hole Effect, Quadratic, and RQuadratic. Up to six models can be used at any one time.
- **Range/Power** refers to two different aspects depending on the chosen variogram model. For all variogram models except power, the value entered refers to the range. This value indicates the lag distance where the range is considered to begin. If the power variogram model is chosen, then the value entered is the power coefficient. A power of one yields a linear model. When you choose multiple models, the range values are summed.
- **Sill/Slope** refers to two different aspects depending on the chosen variogram model. For all variogram models except power, the value entered refers to the sill. This value indicates the lag distance where the sill is considered to begin. If the power variogram model is chosen then the value entered is the slope of the scale coefficient of the curve. When multiple models are selected, the sill values are summed.
- **Angle r**eflects the direction in which the variogram model will be applied. The list shows a selection of different directions that you can use in the semivariogram.
- **Anisotropy** is the ratio between the range in the minor and major directions of the semivariogram.

- **Anisotropy view** diagram shows the directional trends in the data. A wide ellipse indicates that there is a greater degree of correlation of the variances between sample pairs in that direction. Conversely, a narrow ellipse indicates that there is a smaller degree of correlation. The angle setting for each model determines the degree of rotation for each ellipse.



- The **Anistropic Modeling** check box enables you to build more than one model curve for the different directions analyzed. Once you enable this check box, the Angle and Anisotropy settings become available for each chosen variogram model.
- **Suggested Model** analyzes the experimental variogram and chooses the variogram model that best represents it. In some cases, it may not be possible to automatically generate a model. In this case, a warning message appears and you must set a model manually.

The **Nugget** box enables you to force the semivariogram to pass through the y-axis at a higher value. This has a smoothing effect on the kriging process and prevents it from acting as an exact interpolation in scattered areas of high concentration.

Spatial Correlation

Spatial correlation analysis investigates relationships between grids. Because correlation analysis is a statistical technique involving calculation, only numerical grids can be analyzed.

Correlation is a measure of correspondence and is expressed as a coefficient, which is a value between -1 and 1. For positive values, the higher the value, the more closely the grids correspond. For example, a value of 0.85 shows a relatively high correlation, whereas a value of 0.2 shows relatively weak correlation. Negative values show an inverse correlation between grids (for example, if there is strong negative correlation, when the values in one grid are low, the values in another are correspondingly high). For example, a value of -0.85 shows relatively strong negative correlation, whereas a value of -0.2 shows relatively weak correlation.

Correlation Matrix

Using a correlation matrix, you can examine a selection of numeric grids for correlation. All permutations of pairs of grids in the selection are analyzed, and the results are returned in a matrix. The correlation coefficient is displayed for each pair of grids.

Creating a Correlation Matrix

1. From the Vertical Mapper menu, choose the **Data Analysis > Spatial Correlation > Correlation Matrix** command.
2. In the Correlation Analysis dialog box, clear any open grids you do not want to include in the correlation matrix.
3. In the **File Name** box, type a new file name or accept the default.
4. Click the **OK** button.

A Browser opens displaying the correlation matrix.

correlationResults Browser

_#	AvgFamilyIncom	EdHighSchool	EdUnder9	EdUniversity	Employed	Fam_2p	Fam_3p
AvgFamilyIncome	1	0.192161	-0.123606	0.192376	0.603877	0.497405	0.426647
EdHighSchool	0.192161	1	0.426902	0.798454	0.6641	0.707475	0.731512
EdUnder9	-0.123606	0.426902	1	0.399105	0.210033	0.331845	0.3037
EdUniversity	0.192376	0.798454	0.399105	1	0.75267	0.710351	0.739253
Employed	0.603877	0.6641	0.210033	0.75267	1	0.749532	0.828652
Fam_2p	0.497405	0.707475	0.331845	0.710351	0.749532	1	0.674743
Fam_3p	0.426647	0.731512	0.3037	0.739253	0.828652	0.674743	1
Fam_4p	0.499176	0.618836	0.165779	0.644377	0.819851	0.565024	0.801707
Fam_5p_up	0.355345	0.525604	0.235849	0.560581	0.626104	0.409171	0.650757
FamAdult25up	0.355443	0.581323	0.42069	0.59081	0.630656	0.632524	0.669088
FamChildren0_14	0.257137	0.64289	0.324246	0.675724	0.678673	0.495005	0.778177
FamYoungAdult15_24	0.537225	0.615363	0.146401	0.614914	0.814	0.553761	0.814973
HH1p	0.032388	0.0263801	0.357834	0.0845625	0.02098	0.160451	-0.179615
HH2p	0.464524	0.627166	0.308446	0.654596	0.715008	0.922537	0.574764
HH3p	0.446363	0.721452	0.310215	0.757987	0.863759	0.721059	0.946328

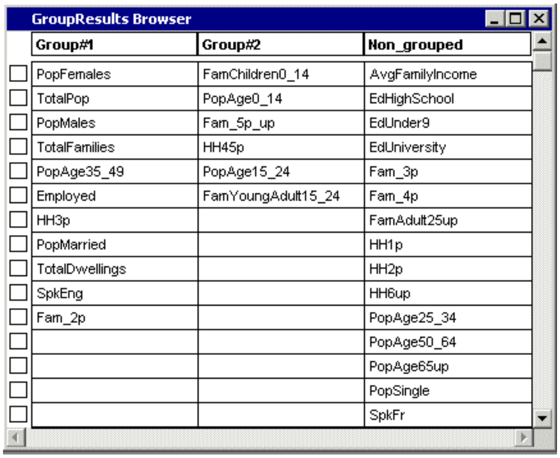
This figure shows part of the correlation matrix for a large number of grids. Note that the value of a grid compared to itself is always 1.

Grouping

Using grouping, you can identify groups of grids that have correlation coefficients with each other above a certain value. This is useful for managing large numbers of grids.

1. From the Vertical Mapper menu, choose the **Data Analysis > Spatial Correlation > Grouping** command.
2. In the Group Analysis dialog box, clear any grids you do not want to be included in the analysis.
3. In the **Group Similarity** box, enter the value you want to use to group the grids. All grids with correlation coefficients above this value will be grouped.
4. In the **File Name** box, type a new file name or accept the default.
5. Click the **OK** button.

When the processing is complete, a Browser opens displaying the number of groups found and the names of the grids in each group. Any grid in a group will correlate with other grids in the group by the value specified in the group similarity box or higher. Any grid that does not correlate with another with at least the specified value will be placed in a group called “Non_Grouped.”


GroupResults Browser

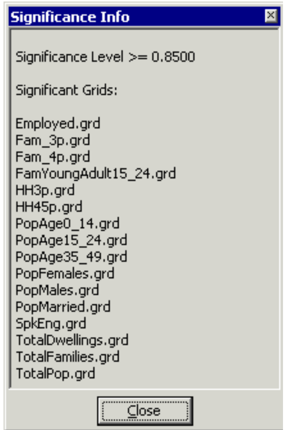
Group#1	Group#2	Non_grouped
PopFemales	FamChildren0_14	AvgFamilyIncome
TotalPop	PopAge0_14	EdHighSchool
PopMales	Fam_5p_up	EdUnder9
TotalFamilies	HH45p	EdUniversity
PopAge35_49	PopAge15_24	Fam_3p
Employed	FamYoungAdult15_24	Fam_4p
HH3p		FamAdult25up
PopMarried		HH1p
TotalDwellings		HH2p
SpkEng		HH6up
Fam_2p		PopAge25_34
		PopAge50_64
		PopAge65up
		PopSingle
		SpkFr

In this example, the specified group similarity was 0.7. The members in Group #1 and Group #2 have a correlation coefficient among themselves of 0.7 or higher.

Significance Analysis

Significance analysis enables you to determine quickly which grids are the most significant by displaying the names of grids that have a weight factor of at least the value you specify. Significance analysis is particularly useful when you have large numbers of grids. It would be possible to achieve the same results manually by creating a component analysis and picking out the grids with the appropriate weights. For more information about principal components analysis, see **Principal Components Analysis on page 164**.

1. From the Vertical Mapper menu, choose the **Data Analysis > Spatial Correlation > Significance Analysis** command.
2. In the Significance Analysis dialog box, clear any open grids you do not want to include in the analysis.
3. In the **Significance Weight** box, enter the minimum weight factor the grids must have to be included.
4. Click the **OK** button. A Browser opens displaying the name of all the grids with at least the specified weight factor.



This Browser shows the results of a significance analysis where the specified weight was 8.5.

Principal Components Analysis

Principal component analysis is an effective and popular method of multidimensional statistical analysis. It provides a method both of reducing the number of variables to be analyzed by eliminating redundancy and of detecting structure in the relationship between variables.

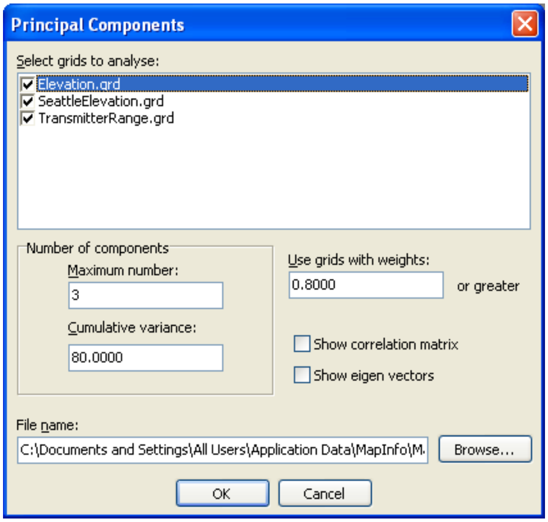
The basic principle of principal component analysis is that several correlated variables can be combined into a single factor. In a very simple example, suppose there is a strong correlation between the presence of a certain mineral in the soil and the presence of gold. The correlation can be summarized in a scatter plot, and a regression line can then be fitted which represents the best summary of the linear relationship between the two variables. If we can define a variable that approximates the regression line, then that variable will capture most of the essence of the two variables. The regression line can be used in future analyses instead of the two variables. Thus, two variables have been reduced to a single factor.

Principal component analysis can be applied where there are multiple components. After the first factor has been extracted, another can be found that defines the remaining variability, and this process can be reiterated. However, as consecutive factors are extracted, they account for less and less variability. It is therefore not particularly meaningful to continue past a certain point.

Performing Principal Components Analysis

1. From the Vertical Mapper menu, choose the **Data Analysis > Spatial Correlation > Principal Components** command.
2. In the Principal Components Analysis dialog box, clear any open grids you do not want to include in the correlation matrix.
3. In the **Number of Components** section, specify the maximum number of components you want to find and the cumulative variance.
4. In the **File Name** box, type a new file name or accept the default.
5. Click the **OK** button.

Exploring the Principal Components Dialog Box



The **Select Grids to Analyze** list displays all open grids. These are enabled by default. You can clear any you do not want to include in the analysis.

The **Maximum Number** box enables you to specify the maximum number of principal components you want to find.

The **Cumulative Variance** box enables you to specify the minimum cumulative variance you want the analysis to achieve.

The **Use Grids With Weights** box enables you to specify the minimum weight factor grids must have in order to be displayed in the output. Specifying any positive value will include grids with negative correlation of at least that value.

The **Show Correlation Matrix** check box gives you the option of displaying a correlation matrix.

The **Show Eigen Vectors** check box gives you the option of generating a grid showing the Eigen values for each of the components.

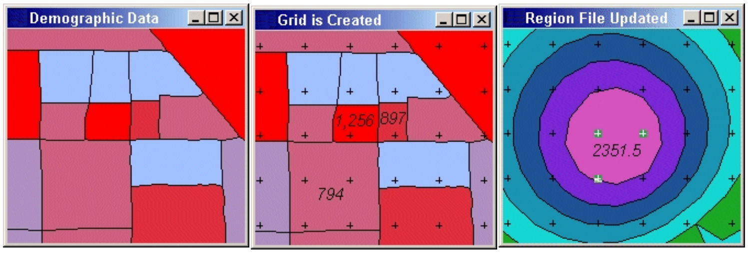
Polygon Overlay

The polygon overlay tool is a grid-based approach to performing a proportional sum analysis between two region files. It is implemented to behave like the Update Column function in MapInfo Professional. A proportional sum analysis is a way of transferring information from one region file into another, taking into account any polygonal overlap between the two files. Generally, most applications involve demographic information from census boundary files. For example, you might update store trade areas with the total population or the average family income.

The process the polygon overlay tool follows can be conceptualized as a three step process. The first step is the vector to raster conversion of the region file that contains the information to be transferred. In most cases, this is demographic data contained in a census boundary file. This step is similar to using the Region to Grid command.

The second step is to calculate the value for each cell in the new grid. The calculation involves knowing how many cells each region contains, and then dividing the region's value by that number. In this way, the region's total value is spread among the grid cells contained inside it. A grid cell is considered to be inside the polygon when the centre of the grid cell (the grid node) is located within the polygon.

The third and final step is to transfer the gridded information to the region file to be updated. A new field is created in the database, and each region in the file is examined against the grid. The sum of the grid values that are contained within each region is then calculated and transferred to the new field in the database.



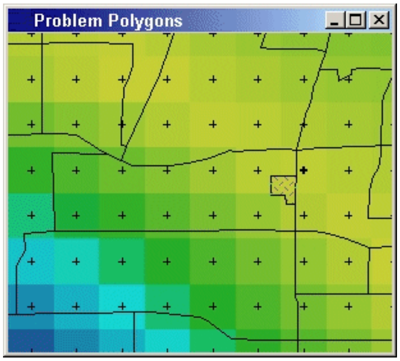
A proportional sum analysis begins generally with demographic information. A grid is then spatially overlaid and grid cells are attributed with a proportional value from the region they fall inside of.

In the second illustration above, the region labeled with a value of 794 has four grid cells that fall inside it where each cell would be attributed with a value of 198.5. In the regions labeled with the values 1256 and 897 there is a single cell contained in the region, and therefore each cell will be attributed with the full value of the region. Once the grid has been processed the cell values are applied to another region file by simply adding the cell values that fall in each region.

In the third figure, the value given to the central circular area will be the sum of the three cells that fall inside. In this case the value will be 2341.5.

The polygon overlay tool makes the assumption that the attribute value of a region is representative of the whole geographic area the region covers and not of a smaller location within the region. When this is assumed, then it is logical to assume that a region's value can be divided and spread equally over the region area. In other words, if you have an area 10 km square with a population of 100 people, it is logical assume that there is one person for every square kilometer of the region. It is important to note that, although this almost never occurs in reality, there is no way of knowing where the population is distributed within the region because of the way the data was collected. Therefore, the best guess is to spread the population evenly throughout the region.

One concern when approaching a proportional sum analysis using a grid-based solution is retaining information from very small polygons when performing the vector to grid conversion step. This becomes an issue when there are no cells in the grid that are contained within the polygon. This is illustrated in the next figure, where the highlighted polygon does not contain a grid cell. In a normal Region to Grid operation, the value associated with this polygon would be lost. However, the Polygon Overlay tool attributes this information to the nearest grid node. For example, in the next figure, the nearest node is to the upper-right of the polygon in question. This does lead to a varying degree of geographic displacement of the information. This displacement will be as large as the cell size used.

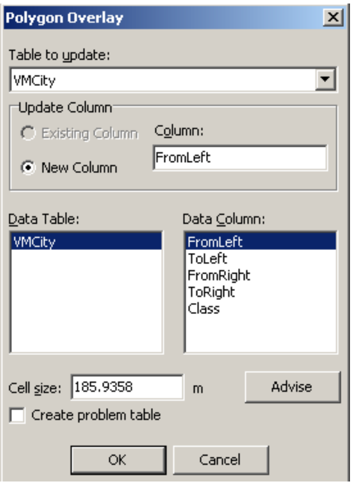


Some polygons may be too small for a grid node to be located inside. In this instance the polygons data is attributed to the closest node. This would be the node to the upper right of the selected region in the above diagram.

This problem can be minimized by decreasing the cell size used in the analysis. You can determine what is a reasonable cell size by clicking the Advise button in the Polygon Overlay dialog box. The polygon file is analyzed automatically to determine how many regions will not have a grid cell located within their boundaries. Note that if you decrease the cell size, longer processing time and more temporary space on your computer system are required. You may also run into situations where there are a certain number of problem polygons, even when the cell size is dramatically decreased. This is symptomatic of sliver polygons in the data set that should be deleted from the file.

After selecting what you think is a reasonable cell size, you can run the analysis using the Create Problem Table option. This will generate a region file of all polygons in the table that do not have a node located inside. If necessary, you can adjust the cell size to reduce the number of problem regions.

Exploring the Polygon Overlay Dialog Box



The **Table to Update** list box enables you to choose the target table that will be updated with the source data.

In the **Column** section, you can choose either of the following options:

Existing Column enables you to choose an existing column in the target table that will be updated with data from the source table.

New Column enables you to specify the name of a new column that will be added to the target table and will contain the data from the source table.

The **Data Table** list displays all the valid tables open in MapInfo Professional. The highlighted table becomes the source data table.

The **Column** list displays the columns available in the selected table, and enables you to choose the column containing the data.

The **Cell Size** box enables you to specify the size of the 32 bit grid that will temporarily hold the source tables data before applying it to the target table.

The **Advise** button Enables you to run an analysis to indicate the number of problem polygons that are found in the target table. These are polygons that are too small to have a single grid cell located inside.

The **Create a Problem Table** option enables you to choose to generate a region table containing a copy of all the problem polygons found in the target table.

Predictive Analysis

Predictive analysis is a method of “smart” classification. It examines the statistical characteristics of multiple input grids found within a given “training” area and then locates other areas with similar properties. For example, oil exploration is one application. Although bore holes give clear results, they are expensive. Suppose experience reveals that the presence of oil in the soil is usually strongly related to the presence of certain minerals or other elements. If we can establish the correlation between the presence of these other elements and oil, then it becomes possible to measure the concentration of the other elements to predict the presence of oil, which is a much less expensive undertaking than drilling new bore holes.

Predictive analysis enables you to “teach” the program how to classify areas based on input criteria. For example, if you identify several regions where oil is present, using these as models, you can teach the program to identify other regions with similar characteristics.

Before running predictive analysis, you must first set up a “teaching table.” This is a MapInfo table containing the region objects you are using as sample regions. It must have at least one integer column, which holds the value of the “class” of region you are defining. In any teaching table, you must define at least two different classes. For example, if you want to identify areas with probable high crime rates based on demographic information, you must define both an area where crime is known to be high, and an area where crime is known to be low. For more accurate results, you can use more than one region for each class type in the teaching table.

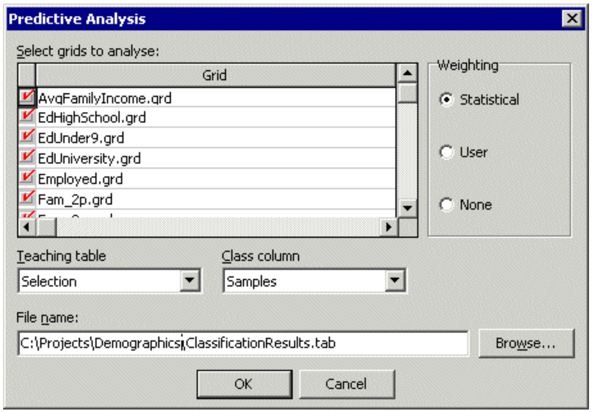
1. Draw MapInfo region objects defining the areas you want to use as samples. You must define at least two regions in different locations within the grid.
2. Save the region objects in a MapInfo table. Ensure the table has at least one column of type integer. Use this column to define the “class” of the region. For example, you might assign the

regions representing a high crime rate “1,” and the regions representing a low crime rate “2.” Any region assigned a value of 0 in this column will not be included in the analysis.

This table is the “teaching table.” The colours that you assign to the regions will be used for displaying the results.

3. From the Vertical Mapper menu, choose the **Data Analysis > Predictive Analysis** command.
4. In the Predictive Analysis dialog box, clear any grids you do not want to include in the analysis.
5. From the **Teaching Table** list, choose the name of the table that contains the region objects you want to use for your analysis.
6. From the **Class Column** list, choose the name of the column that contains the integer value defining the “class” of the region.
7. In the **Weighting** section, choose one of the following:
   - **Statistical** The grids are weighted automatically according to statistical calculations.
   - **User** Enables you to specify how grids should be weighted; when you choose this option, a new column is added to the list of grids where you can enter the weight of each of the grids.
   - **None** No weighting is applied; the grids are treated as having equal importance.
8. In the **File Name** box, type a new file name or accept the default.
9. Click the **OK** button.

Exploring the Predictive Analysis Dialog Box



The **Select Grids to Analyze** list enables you to clear any open grids you do not want to be included in the analysis.

The **Weight** section enables you to choose a method for weighting. These options are available:

Statistical the grids are weighted automatically according to an appropriate statistical calculation.

User enables you to specify how grids should be weighted; when you choose this option, a new column is added to the list of grids enabling you to specify the weight.

None no weighting is applied; the grids are treated as having equal importance.

The **Teaching Table** list enables you to choose the table containing the regions to use for teaching.

The **Class Column** list enables you to choose the column in the table that contains the class of the region; the class must be an integer.

The **File Name** box enables you to enter a new file name.

10

Aggregating Data

This chapter describes two methods of data aggregation: simple point aggregation and point aggregation with statistics.

This chapter covers:

- when to aggregate data
- how to aggregate data using simple point aggregation
- how to aggregate data using point aggregation with statistics

In this chapter:

Aggregating Data

Data aggregation is a mathematical process for reducing the number of points in a point file. Different methods are available to process a point file, but the underlying goal is to spatially group and statistically merge points that are in close proximity. This not only results in fewer points but also creates a more uniform distribution of points and point values.

Vertical Mapper provides two options for data aggregation.

- Simple Point Aggregation quickly aggregates data when you do not need to keep track of the spatial or mathematical properties of the aggregation process at each point;. For example, knowing the average of the aggregated points is sufficient.
- Point Aggregation With Statistics tracks the statistical parameters associated with each aggregated point (for example, standard deviation or coefficient of variation). This technique can also produce a region file displaying the actual circular or square cell region used to gather and aggregate points. Statistical information is attached as attributes to both the new point file and to the region file.

When to Aggregate Data

The following reasons for aggregating data are the most common ones.

- There are multiple values at a given location.
- Data is erratic, and the average of the data area represents an appropriate estimate of a point.
- The sum of the data in a given area is required.
- A representative value at an “average” location is acceptable when the location of the data is not accurate.
- The original data set has too many points to permit efficient processing. The number of points can be substantially reduced without adversely affecting the accuracy of any operation.
- Values representing specific spacing or area criteria applied over the entire mapped area (most applicable to the “square cell” method) are required.

Techniques for Data Aggregation

The following data aggregation techniques are available:

- Simple Point Aggregation
- Point Aggregation with Statistics, which comprises the Forward Stepping, Cluster Density, and Square Cell aggregation techniques.

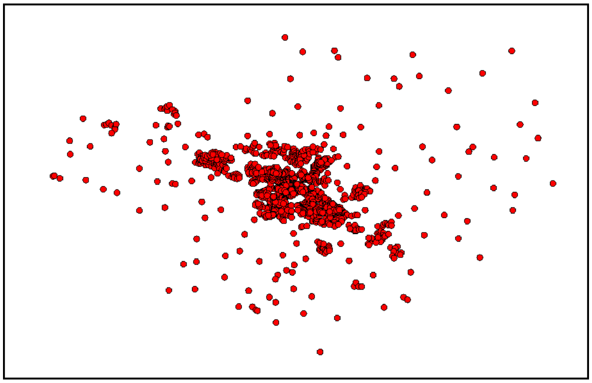
When you use these techniques, Vertical Mapper creates a new file containing the aggregated points and leaves the original point file unaltered.

Simple Point Aggregation

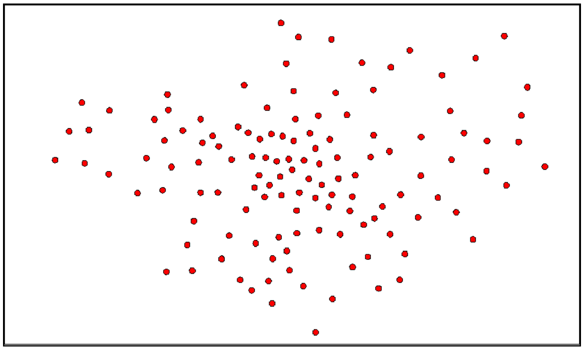
Simple point aggregation is useful for quickly grouping points that are virtually coincident or that are separated by relatively small gaps. For example, when soil samples are collected, you may need to take several samples at each sample site. When creating a surface of this data, you may want to average all of the samples taken at each site before proceeding with the surface creation. It may also be desirable to aggregate data that has poor reproducibility over a large area and achieve a more smoothing or averaging effect.

Simple point aggregation is also used as a preliminary data smoothing technique for TIN creation in the Triangulation with smoothing interpolation method, for natural neighbour region creation in the natural neighbour interpolation method, and in the kriging interpolation. Here, highly variable data points spaced closely together can be aggregated and new values calculated using a statistical expression. For example, if two points are very closely spaced, even a small difference in z-value will generate a steep slope between them. This slope affects the interpolated surface for a significant distance beyond the two points, creating unwanted rises and dips.

Simple point aggregation uses a coincident point distance value for spatially aggregating points. This means you need to specify how close the points will be before they are considered coincident. The result of simple point aggregation is shown in the next two figures.



Original distribution of points.



Distribution of points after simple point aggregation.

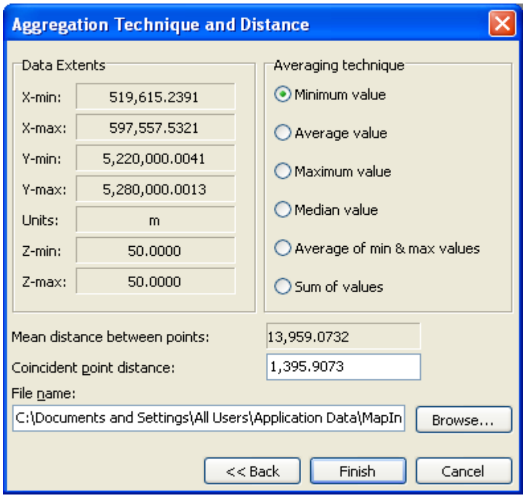
Performing Simple Point Aggregation

You can use simple point aggregation to quickly group points that are virtually coincident or that are separated by relatively small gaps.

1. From the Vertical Mapper menu, choose the **Data Aggregation > Simple Point Aggregation** command.
2. From the **Select Table to Aggregate** list in the Select Table and Column dialog box, choose a point table.
3. From the **Select Column** list, choose a column of data to transfer to the aggregated file.
4. Enable the **Ignore Records Containing Zero** check box to include only non-zero records.
5. Click the **Next** button.
6. In the Aggregation Technique and Distance dialog box, choose an averaging technique.
7. In the **File Name** box, type a new file name or accept the default.
8. Click the **Finish** button.

Exploring the Aggregation Technique and Distance Dialog Box

The left side of the Aggregation Technique and Distance dialog box summarizes the spatial characteristics of the point file including the file extents, the z-value range, and the coordinate system units. The right side of the dialog box contains settings that control the statistical processing of the passed values.



The **Averaging Technique** section enables you to choose one of six statistical methods to determine the z-value of the aggregated points.

- **Minimum Value**: The minimum value is the lowest value within the “same point” distance of the aggregated coordinate.
- **Average Value**: The average value is determined by averaging all valid points within the “same point” distance of the aggregated coordinate.
- **Maximum Value**: The maximum value is the highest value within the “same point” distance of the aggregated coordinate.

- **Median Value**: The median value is the middle value of all points within the “same point” distance of the aggregated coordinate. If there are an even number of points, the median value is the average of the two middle values.
- **Average of Min & Max Values**: This value is determined by averaging the minimum and maximum values of all points within the “same point” distance of the aggregated coordinate.
- **Sum of Values**: The sum of values is determined by summing all values of valid points within the “same point” distance of the aggregated coordinate.

The **Coincident Point Distance** box—enables you to define the minimum distance between two points before they are considered coincident.

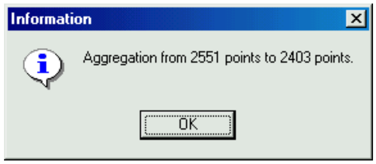
The default value is 10 percent of the mean distance between the data points and is measured in coordinate system units. Choose a small or zero distance to treat points in the data set that are very closely spaced or overlie each other. As the distance setting becomes greater, there will be more coincident points, and consequently the number of points in the output data set will decrease.

Point Aggregation with Statistics

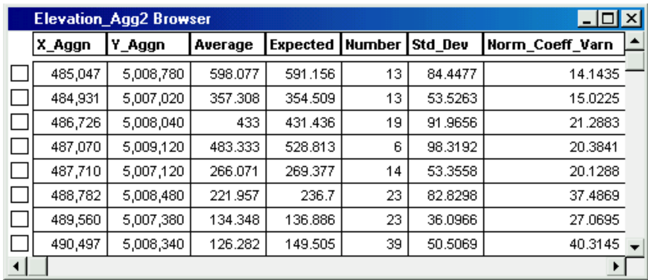
The point aggregation with statistics techniques provide information about the mathematical and spatial characteristics of the aggregation process. To quantitatively measure the aggregation process, Vertical Mapper offers three techniques: Forward Stepping, Cluster Density, and Square Cell. These methods are grouped under the category Point Aggregation With Statistics.

1. From the Vertical Mapper menu, choose the **Data Aggregation > Point Aggregation With Statistics** command.
2. From the **Select Table to Aggregate** list in the Select Table and Column dialog box, choose a point table from the list of open MapInfo tables.
3. From the **Select Column** list, choose a column of data to transfer to the new aggregated file.
4. Enable the **Ignore Records Containing Zero** check box to include only non-zero records, and click the ***Next*** button.
5. To aggregate coincident points, clear the **No Coincident Point Handling** check box in the Select Coincident Point Technique dialog box.
6. Choose an averaging technique option, and click the **Next** button.
7. In the **Aggregation Technique** section of the Select Aggregation Technique and Statistics dialog box, choose an aggregation technique.
8. In the **Aggregate Point Attributes** section, enable any of the check boxes.
9. In the **Aggregation Distance** box, type the desired distance.
10. Enable the **Create Regions Table** check box if you want to build a MapInfo table of regions, and the **STD Ellipse Table** check box if you want to build a table of standard deviation ellipses.
11. In the **File Name** box, type a new file name or accept the default.
12. Click the **Finish** button.

Once the process is complete, a dialog box appears stating the extent of the aggregation as shown below. The aggregated point file appears in a new Map window with a default symbol style applied to each point. If you choose the Create Regions Table option, the aggregation regions table appears in a separate Map window and is assigned the suffix “AggRegion”.



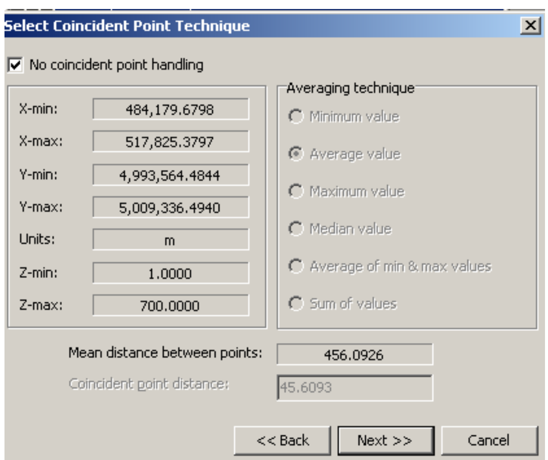
The calculated statistical information is retained in the Browser window of both the aggregated point file and the aggregation regions file as shown below.


Elevation_Agg2 Browser

X_Aggn	Y_Aggn	Average	Expected	Number	Std_Dev	Norm_Coeff_Varn
485,047	5,008,780	598.077	591.156	13	84.4477	14.1435
484,931	5,007,020	357.308	354.509	13	53.5263	15.0225
486,726	5,008,040	433	431.436	19	91.9656	21.2883
487,070	5,009,120	483.333	528.813	6	98.3192	20.3841
487,710	5,007,120	266.071	269.377	14	53.3558	20.1288
488,782	5,008,480	221.957	236.7	23	82.8298	37.4869
489,560	5,007,380	134.348	136.886	23	36.0966	27.0695
490,497	5,008,340	126.282	149.505	39	50.5069	40.3145

Exploring the Select Coincident Point Technique Dialog Box

This dialog box is the same one that is used to set parameters for simple aggregation. There are two steps in statistical aggregation. The first step allows you to deal with virtually coincident points separately before you aggregate the points further. For instance, you may want to average all the soil samples collected at each site before you average and sum all the soil sample sites that occur in a given unit area.

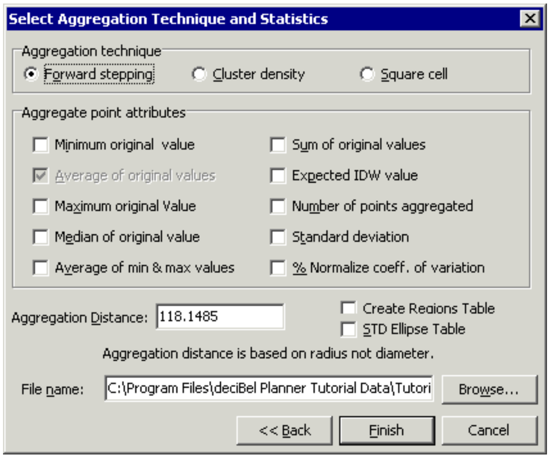


The **No Coincident Point Handling** check box is enabled by default, and the entire dialog box is greyed out.

In most cases, you will not need to aggregate virtually coincident points before you proceed with statistical aggregation, and the settings in this dialog box will be ignored. This is not true when you employ cluster density aggregation. With cluster density, coincident point handling is always performed regardless of whether the dialog box is greyed out. If you do not choose any of the settings in this dialog box, the default values will be used. A reminder will appear when this technique is chosen.

Exploring the Select Aggregation Technique and Statistics Dialog Box

The Select Aggregation Technique and Statistics dialog box enables you to set the aggregation technique and aggregate point attributes.



The **Aggregation Technique** section—enables you to choose one of the three aggregation techniques: forward stepping, cluster density, and square cell.

The **Aggregate Point Attributes** section—enables you to choose the following statistical parameters, computed during the aggregation process, that will be assigned to the aggregated point file and region file (if chosen).

- **Minimum Original Value**: The minimum value selected for aggregation.
- **Average of Original Value**: The average value of all the points selected for aggregation. This value must always be assigned to the new aggregated tables.
- **Maximum Original Value**: The maximum value selected for aggregation.
- **Median Original Value**: The median value is the middle value of all points selected for aggregation. If there is an even number of points, the median value is the average of the two middle values.
- **Average of Min & Max Value**: This value is the average of the minimum and the maximum values.
- **Sum of Original Values**: The value obtained by summing all values selected for aggregation.
- **Expected IDW Value**: The value is obtained by summing the weighted value for each point selected for aggregation and dividing this value-weighted sum by the sum of the weights.

  The weight associated with each value is inversely proportional to the square of the normalized distance between the value and the reference location. The normalized distance is the actual distance from the reference point divided by 1.01 times the “same point” distance. Therefore, values close to the reference point will have more influence than values farther from the reference point. The inverse distance weighted value is considered to be a reasonable estimate of the value at the aggregated coordinate.

- **Number of Points Aggregated**: This value is the total number of points selected for aggregation. It does not include the points processed in the Coincident Point Handling step.
- **Standard Deviation**: This value is the degree of dispersion between the mean of the values for those points selected for aggregation.
- **% Normalized Coeff. of Variation**: The coefficient of variation is the standard deviation divided by the average expressed as a percentage (multiplied by 100).

  The value is dimensionless and indicates data dispersion, especially when the average is not close to zero. A “normalized” coefficient of variation is calculated using the standard deviation divided by the difference between the average and the minimum value of the complete data set, not just the data in the local aggregated region. The coefficient can still be large when the average value is close to the minimum value, but the normalized coefficient is representative of the original range of values and not their absolute value.

The **Aggregation Distance** box—enables you to specify the distance used to group points for aggregation.

This distance is defined differently for each of the three aggregation techniques. For the Cluster Density and Forward Stepping techniques, aggregation distance is defined by the radius of a user-specified circular search area centred on each aggregation cell. For the Square Cell technique, Aggregation Distance is defined by the width of the square aggregation cell.

The **Create Regions Table** check box—enables you to build a table of the regions used to group the point data selected for aggregation.

This option enables you to visually inspect the results of the aggregation process. The order in which these regions are created is the same order in which the point file was aggregated. By opening a Browser window of the region file and choosing each record in the list one at a time, you can see how the regions are processed in the Map window.

Note You can use the created regions to produce a MapInfo Professional coloured thematic map where each region is thematically shaded according to one of the computed statistical values.

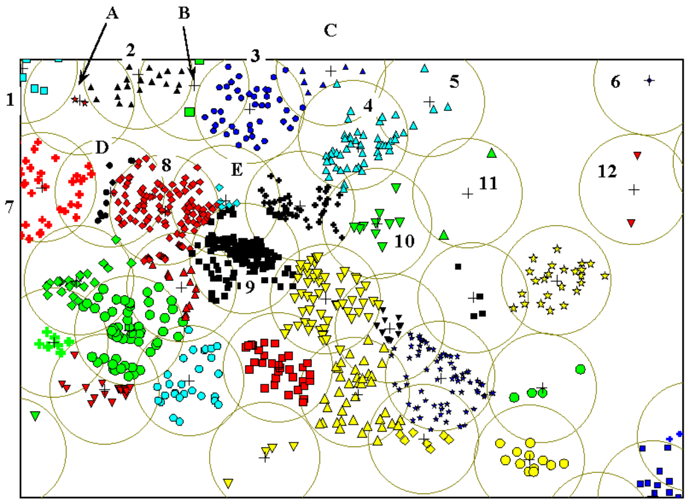
The **Create STD Ellipse Table** check box—enables you to build a table of standard deviation ellipses. For more information, see **Building a Table of Standard Deviation Ellipses on page 181**.

Forward Stepping Aggregation

Due to its speed and effectiveness, the Forward Stepping Aggregation technique is an appropriate method for any general aggregation application. This technique is useful when your data has a truly random distribution or when the other two techniques are not appropriate.

This aggregation process aggregates points by moving through the data set from left to right and then top to bottom. The process begins by sorting the data points into rows that are generally three times the aggregation distance. This sorting is performed to determine where the aggregation will begin each left-to-right swath. There are no settings you can use to alter this process. Beginning with the data points in the upper left (northwest) corner of the data set, a circular search radius is created as specified by the Aggregation Distance setting. All data points that fall inside this search radius are selected and flagged. This prevents these data points from being aggregated to another location. The geocenter of the selected points is then determined. This becomes the location of the new aggregated point. The process ends by performing the aggregation calculations on the selected points as specified in the Point Aggregation dialog box, and the results are attributed to the new aggregated point.

Once the first point is processed, the procedure sweeps from left to right and top to bottom across the study area, selecting and aggregating unflagged points. It is important to note that not every data point will be aggregated on the first pass through the data set. Normally a second pass is required to aggregate those points missed on the first pass. The results are shown in the next figure.

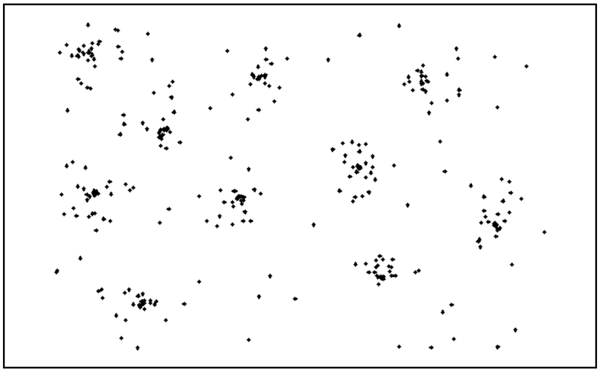


Example of circle aggregation using the Forward Stepping Aggregation technique. The shaded points are the original data points. The crosses represent the newly aggregated points, and the circles represent the aggregation region. The original points have been coded to show to which aggregation region they belong.

In the figure above, you may notice that inappropriate aggregation decisions have been made in certain locations of the point file as well as the degree of overlap of the aggregation regions. In the upper left corner of the diagram there are two examples of an inappropriate aggregation, marked by the letters A and B. In both cases you would aggregate these points differently if you performed this process manually. The reason these points are aggregated this way has to do with the two aggregation passes this technique performs; the second pass aggregates the remaining unflagged points, resulting in a large degree of overlap of the aggregation regions. Some of the aggregation regions in the above diagram have been numbered to show the aggregation process order. The letters show which points were aggregated on the second pass.

Cluster Density Aggregation

The Cluster Density Aggregation technique is typically used when a visual clustering effect occurs in the dispersion of the data points. For example, demographic data representing rural areas may often exhibit a ‘shotgun’ pattern (see the next figure). Data from each small community is considerably more densely distributed than in the surrounding countryside. Cluster density does, however, process large data sets more slowly than the other aggregation techniques.

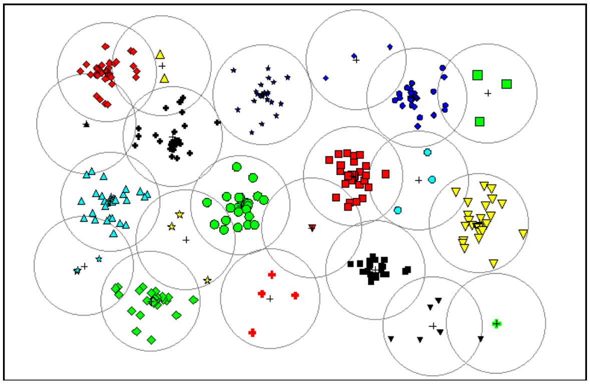


Original distribution of data points.

The Cluster Density Aggregation technique is usually most effective for small to medium size data sets, because it makes better decisions during the aggregation process. It aggregates the more densely clustered points first within a reasonable processing time. However, it is not appropriate if you need to know the number of points aggregated or if there are coincident points in the data. Coincident points are always aggregated first and are not included in the statistics appended to the new point file. The alternative is to use the Forward Stepping Aggregation technique.

The Cluster Density Aggregation technique looks at the entire data set prior to aggregation and determines the single most densely populated area that would fall inside the user-specified search radius (aggregation distance). Points that fall inside the search radius are chosen and flagged. The geocenter of these points is calculated and that position becomes the location of the new aggregated point. Calculations are performed on the values of the selected points (as specified in the Point Aggregation dialog box) and the results are attributed to the new geocentred point. Then, the area with the second highest density of points is chosen and the process is repeated. At each stage, the entire remaining data set must be examined for its density patterns to avoid using previously aggregated points and to factor in the removed points in the density analysis.

Cluster density aggregation always includes coincident point handling regardless of whether or not you request this type of handling, and the point density calculation used in the aggregation technique does not handle points that are coincident. If you do not choose any of the settings on the Coincident Point Handling dialog box, the default values will be used. A warning message appears reminding you when this technique is chosen.



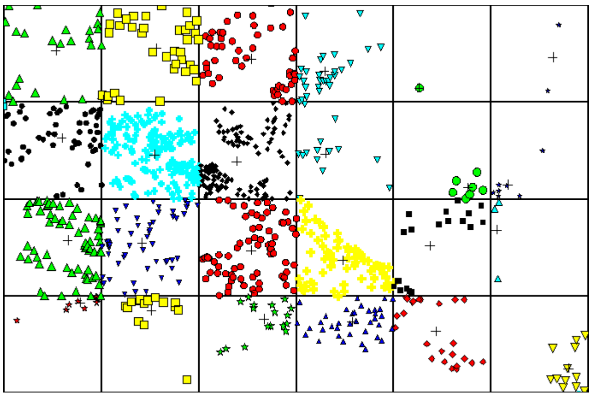
An example of circle aggregation using the Cluster Density Aggregation technique.

There are a total of 330 points in the sample data set shown in the previous figure. After using the cluster density aggregator, you can see that the data is clustered in about ten separate areas. The more randomly distributed points lying outside these clustered zones (circles) are also aggregated, but with significantly fewer points included. Although many circles overlap, the degree of overlap is significantly less than when the forward stepping method is used.

Square Cell Aggregation

The Square Cell Aggregation technique is typically used when values are required that represent specific areas, for example, a density map of the number of new housing units per square kilometre, or when you want to avoid overlapping aggregation regions.

This method divides the area covered by the point file into adjacent squares determined by the aggregation distance. The points that fall inside any of these squares are aggregated to a new point created at the geocentre of the aggregated points (not at the centre of the square). As with the previous two techniques, the specified statistical information is then attached as attributes to the new aggregated point.



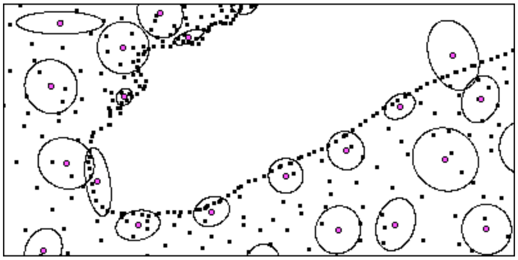
An example of data points aggregated using the Square Cell Aggregation technique.

Although there is no overlap of the aggregation regions in the figure above, points have been aggregated inappropriately in several areas. Therefore, the best results require a certain degree of overlap.

Building a Table of Standard Deviation Ellipses

Standard deviation is a measure of dispersion in point patterns. Typically, it measures dispersal in terms of a circle around the mean centre. The circular model, however, takes no account of the fact that spread may be different in different directions.

The standard deviation ellipse summarizes dispersion in a point pattern in terms of an ellipse rather than a circle. The ellipse is centred on the mean centre, with its long axis in the direction of the maximum dispersion and its short axis in the direction of the minimum dispersion. The axis of maximum dispersion in a point pattern is always at right angles to the axis of minimum dispersion. The length of the X axis is set to one standard deviation of the X values, and the length of the Y axis to one standard deviation of the Y values.



An example of standard deviation ellipses. The ellipses are generated around the original data points, which are at the mean centres of the ellipses containing them. The X axis extends one standard deviation of the X values, and the Y axis extends one standard deviation of the Y values.

11

Using Natural Neighbour Analysis

This chapter describes how to perform natural neighbour analysis. Using the Natural Neighbours technique, you can analyze point data that needs to be mapped to discrete regions with constant values assigned to each point.

This chapter covers:

- natural neighbours
- how to create regions from points
- how to calculate a region area

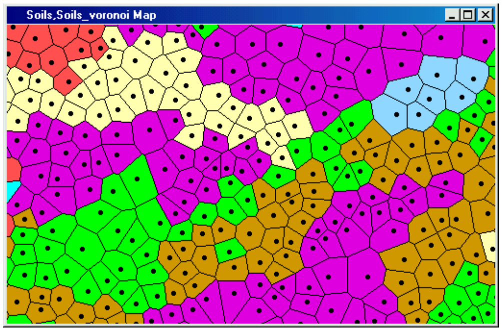
In this chapter:

Understanding Natural Neighbours

While the strength of Vertical Mapper lies in the ability it gives you to create a continuous grid from non-continuous data points, not all types of data are best represented as a continuously varying surface. Some types of point data should be mapped as discrete regions within which the values assigned to each point are constant. Point data such as this is referred to as having a natural neighbour. Examples include store locations that have natural areas of influence and point observations that record classified values (for example, land use).

In Vertical Mapper, natural neighbours are built around data points using Delaunay triangulation. A network of Thiessen polygons is generated from the point locations, creating what is called a Voronoi diagram. As shown below, a Voronoi diagram is a network of MapInfo regions. Any location within a region will always be closer to the enclosed point in that region than to the enclosed point of any other region. Each site exerts a natural area of influence in relation to adjoining sites.

Assigning all the attribute fields from the original point data base to the new region file is a unique feature of Vertical Mapper. Once the natural neighbours have been mapped, you can visually analyze and compare any column of information attached to the region table using the MapInfo Professional thematic mapping tools.



A Voronoi diagram showing the natural neighbours of a series of point observations. One of the columns of data assigned to the region table has been thematically mapped.

The same natural neighbour relationships that are used to examine the distribution and proximity of point occurrences over space can also be applied to quantify point density. Of common concern through a wide range of disciplines is the examination of the spatial occurrence of certain phenomena. Examples that reflect the extensive range of studies covering such phenomena include the location of towns in a state (geography) or seismic events over a continent (geology). Point distribution over space can be observed qualitatively simply by plotting the data on a map. However, producing a quantitative map of the distribution requires some analysis of the relative proximity of the points in relation to one another.

Computing a natural neighbour (Voronoi) diagram for all points yields an excellent measurement of point density. A representative density surface grid can be produced by calculating the area of the natural neighbour region encompassing each point, attaching that area as an attribute to the point, and generating a grid of the new point file through interpolation. This function has been automated in Vertical Mapper and is a powerful tool in natural neighbour analysis.

The following section describes the two main procedures in Vertical MapperSurface Analysis that make use of natural neighbour relationships: building Voronoi diagrams from point tables and calculating point density.

Creating Regions from Points (Voronoi)

Using the Voronoi technique, you can generate a region around each individual data point. The resulting network of regions is often referred to as a Voronoi diagram.

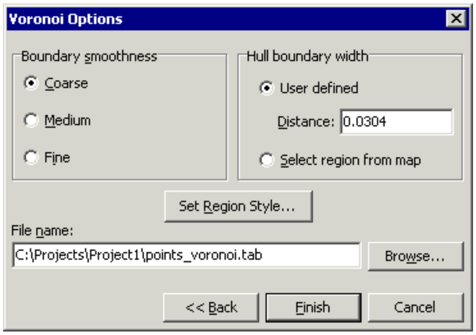
Before running to completion, the point table is checked to ensure that no coincident points are present. If coincident points are found, an error message will appear warning you that the input table contains coincident points. You can edit the table or use data aggregation tools to fix this problem. The extended error code contains the row ID of the first coincident point found.

1. From the ***Vertical Mapper*** menu, choose the **Natural Neighbour Analysis >Create Regions From Points (Voronoi)** command.
2. From the **Create From Table** list, choose a table of points from the list and click the **Next** button. All data fields present in the point table are assigned to the new Voronoi region table.
3. In the Voronoi Options dialog box, choose the required settings to control the manner in which the Voronoi diagram is created.
4. Click the **Finish** button.

   The Voronoi regions table opens in a new Map window.

Exploring the Voronoi Options Dialog Box

The most critical area of the Voronoi diagram is the outer margin where no points are present to control the formation of the outermost polygons.

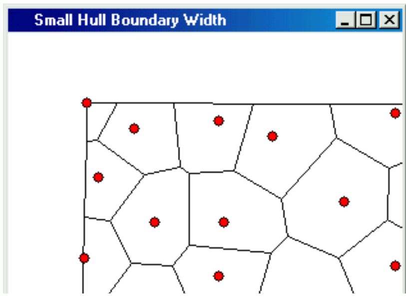


The **Boundary Smoothness** section—enables you to determine the number of line segments that are used to construct the corners of the outer hull of the diagram. The finer the setting is, the smoother the corners will become.

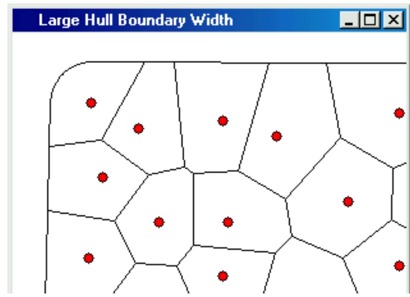
The **Set Distance** box in the **Hull Boundary Width** section—enables you to set the distance, in map units, of the outermost polygon edge from the outer points. Since no points are present beyond the margin that controls the creation of polygons, this setting restricts the construction of polygon sides to a fixed distance from each outermost point (see the next two figures).

The **Select Region From Map** check box—enables you to use a pre-defined MapInfo region as the Voronoi boundary.

The **Set Region Style** button—enables you to customize the fill and line style pattern of the Voronoi diagram.



In this Voronoi diagram, the Hull Boundary Margin width setting is very small (1). Therefore polygons are effectively cut off at the outermost limit of the point file.



For the same data, the Hull Boundary Margin width has been set to 1 000, resulting in the construction of outer polygon edges 1 000 metres away from the outermost points. The rounded corners of the outer boundary are controlled by the smoothness setting.

Calculating the Region Area

The calculate region area command provides a quick way of seeing how densely packed points are. It adds a column to the original data set specifying the area of the Voronoi region of each point. The closer together points are, the smaller the area of the Voronoi region will be.

1. From the Vertical Mapper menu, choose the **Natural Neighbour Analysis > Calculate Region Area** command.
2. In the Region Area dialog box, choose the appropriate table of points that contains the data to be processed in the **Create From Table** list and click the **Next** button.
3. Choose one of the following options from the **Boundary Smoothness** section:
   - Coarse
   - Medium
   - Fine
4. Choose one of the following options from the **Hull Boundary Width** section:
   - **User Defined**—enables you to enter a value in the Width box.
   - **Select Region From Map**—enables you to use the VM Picker tool to choose the hull boundary.
5. Click the **Finish** button.

   The region area is added as a column to the grid table and displayed in a new browser window.

12

Creating 3D Views Using GridView

GridView enables you to generate 3D renderings of one or more numeric grid files as well as adjust many settings such as the lighting, viewing angles, reflectance, and shading properties. Through the use of drape files, text, line work, colour-filled polygons, and raster imagery, you can render scenes in three dimensions. In addition, you can save any scene as a bitmap image with a corresponding MapInfo .tab file. This allows you to incorporate 3D scenes within a MapInfo Professional Layout window.

In this chapter:

Launching GridView

GridView can be accessed in three different ways. The most common way is through the Grid Manager. The highlighted grid in the Grid Manager will be the master grid in the resulting 3D scene.

You can also access GridView by clicking the GridView button in the Point-to-Point Visibility dialog box, available from the Point-to-Point Visibility button on the Vertical Mapper toolbar. The advantage of launching GridView from the Point-to-Point Visibility dialog box is that you can choose the exact viewpoint locations used in rendering the 3D scene. For more information on the Point-to-Point Visibility tool, see **The Point-to-Point Visibility Function on page 129**.

In addition, you can launch GridView by running the Gridview.exe file located in the Vertical Mapper program folder. Open a grid from the File menu in GridView once the program has initialized.

The master grid is used to decide which other grids can be loaded in the scene and is the grid that all scene viewing parameters are based on. The distance units used in many of the settings are also based upon the projection system of the master grid. For example, if the master grid is in a UTM projection, then the viewing parameters will be in metres.

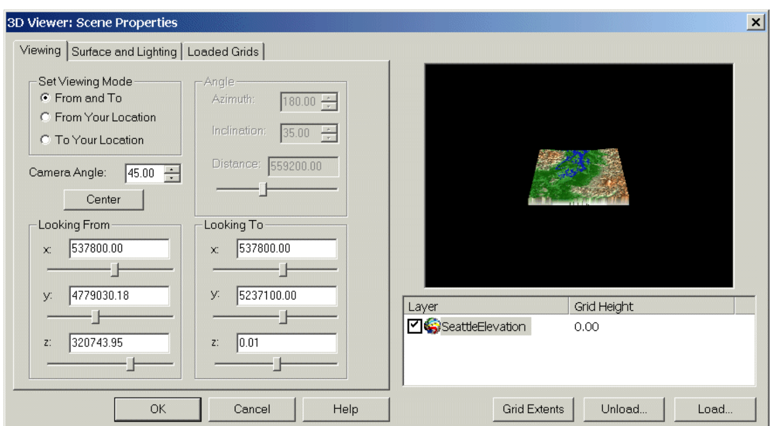
- In the Grid Manager, click the **3D View** button and choose the **Run 3D Viewer** command.

Exploring the Scene Properties Dialog Box

Once GridView has been launched, the 3D Viewer: Scene Properties dialog box appears. This dialog box contains three tabs: Viewing, Surface and Lighting, and Loaded Grids. A preview window provides real-time updated views of the rendered grid file, immediately showing the effect of changing the various settings. Also, the Grid Layer Control helps manage the multiple grids and drape files that can be rendered within a scene. See **The Grid Layer Control on page 199**.

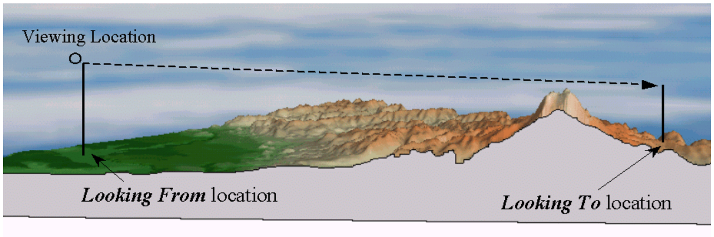
The Viewing Tab

The Viewing tab enables you to set viewing parameters. It contains settings for determining the view perspective of a rendered scene.



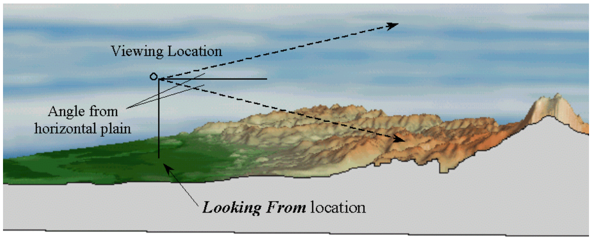
The **Set Viewing Mode** section enables you to determine the location of the viewing position in geographic space. There are three ways in which this location can be defined and, depending on which mode is chosen, different settings will be available on the Viewing tab. The default is the From and To mode.

- The **From and To** mode enables you to specify the x- and y- coordinates of where the viewer is standing in the scene and the x- and y- coordinates of the location the viewer is looking towards. You can also specify how far above or below the grid surface these two locations are. In other words, you specify the x-, y- and z- coordinates of where the viewer is looking from and the x-, y- and z- coordinates of where the viewer is looking towards (see the next figure). These locations are entered into the Looking From and Looking To sections of the Viewing tab. Values must be expressed in the units determined by the coordinate system of the master grid.



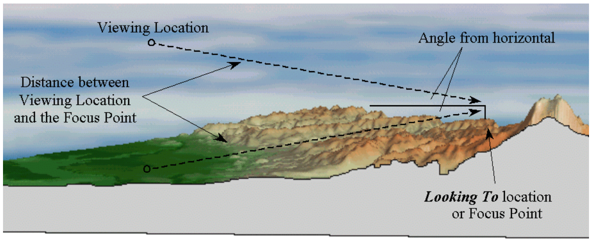
This diagram illustrates the From and To viewing mode.

The **From Your Location** mode requires you to specify the x- and y-coordinates of where the viewer is standing, as well as how far above or below the grid surface this location is. Also, you will need to specify the direction in which the viewer is looking with respect to true north and the angle from the horizon plain at which the scene is being viewed (see the next figure). The x-, y- and z- coordinates are specified in the Looking From section of the dialog box, the view direction is specified in the Azimuth setting, and the horizon angle is specified in the Inclination setting. These settings are found in the Angle section of the Viewing tab.



This diagram illustrates the From Your Location viewing mode.

- The **To Your Location** mode requires you to specify the x- and y-coordinates of where the viewer is looking towards (focus point), how far above or below the grid surface this location is, the angle from the horizontal plain at which the scene is being viewed, and the distance between the viewing location and the location on the grid that is being viewed (see the next figure). The x-, y- and z- coordinates are specified in the Looking To section of the dialog box, the viewing direction is specified in the Azimuth setting, and the angle is specified in the Inclination setting. These settings are found in the Angle section of the Viewing tab.



This diagram illustrates the To Your Location viewing mode.

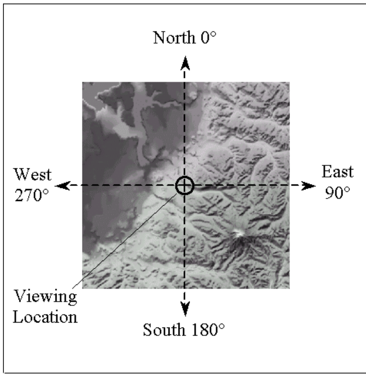
The **Camera Angle** box enables you to set the width of the view-scene in degrees. It allows you to control the field of view similar to using a wide-angle lens on a camera. Using a smaller angle will result in seeing less of the grid.

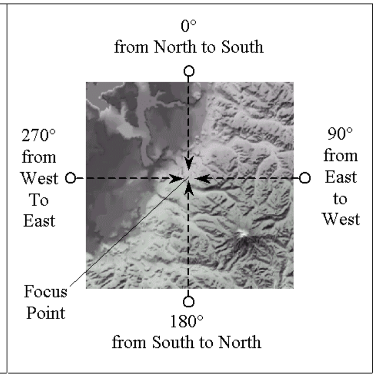
The **Center** button enables you to restore all default viewing settings.

The **Looking From** section defines the x-, y- and z- coordinates of the location that the viewer is looking from. The x, y coordinate pair specifies the surface position, and the z-value specifies the distance above or below the surface elevation. These values must be expressed in the units of the coordinate system being used by the master grid. The default x-coordinate corresponds to the geographic centre of the master grid. The default y-coordinate defaults to 2.5 times the horizontal distance of the master grid. The default z-coordinate is the lowest value in the master grid. You can modify these settings by entering a new value in the edit dialog box or by moving the slider bar to the left or right. When the slider bar is in use (indicated by the dashed line around it), the arrow keys can be used to modify this value.

The **Angle** section contains view settings. Initially this section is unavailable because the default is the From and To viewing mode.

- **Azimuth** is the angle in the x, y plane at which you view the grid. Depending on the viewing mode, this setting will behave differently. For example, when the From and To mode is chosen, the Azimuth setting is not available. When you choose the From Your Location mode, the azimuth is the direction the viewer is looking to, with respect to true north, from the viewing location. Therefore, a value of zero degrees will rotate the viewing direction so that the viewer is looking north. Likewise, a value of 90 degrees will rotate the viewing direction to the east, 180 degrees to the south, and 270 degrees to the west. This is illustrated in the next figure. When the To Your Location mode is chosen, the azimuth is the direction the viewer is looking toward, with respect to true north, in relation to the focus point. The focus point is the location on the grid that is being viewed and is specified by the Looking To setting. Therefore if the looking to location is the centre of the grid, then a value of zero degrees will place the viewer in the north looking south. Likewise, a value of 90 degrees places the viewer in the east looking west; a value of 180 degrees places the viewer in the south looking north; and a value of 270 degrees places the viewer in the west looking east. This is illustrated in the next two figures.





In the From Your Location mode, the azimuth is the direction the viewer is looking toward in relation to the viewing location.

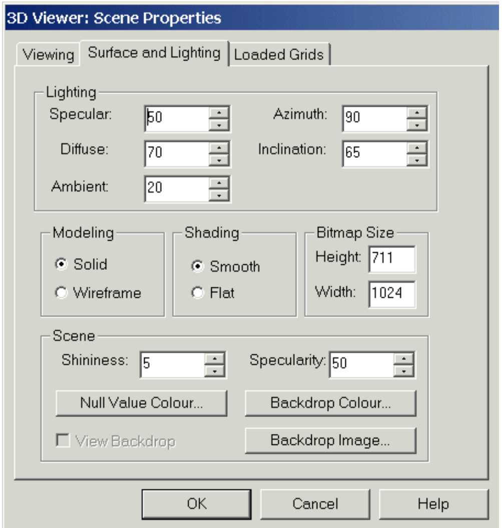
In the To Your Location mode, the azimuth is the direction the viewer is looking toward in relation to the focus point.

- **Inclination** is the angle measured from the horizon to the line of sight at either the viewing location or the focus point, depending on the selected viewing mode. If the From and To mode is chosen, the Inclination setting is applicable. When the From Your Location mode is chosen, the inclination is the angle from the horizon to the line of sight at the viewing location. When the To Your Location mode is chosen, the inclination is the angle from the horizon to the line of sight at the focus point.
- **Distance** is the distance between the viewing location and the focus point and is expressed in the coordinate units of the master grid. You can modify this value by entering a new value in the edit dialog box or by moving the slider bar to the left or right. When the slider bar is in use (indicated by the dashed line around it), the arrow keys can be used to modify this value. This setting is only available for the To Your Location viewing mode .

The **Looking To** section defines the x-, y- and z- coordinates of the location the viewer is looking towards. The x, y coordinate pair specifies the surface position, and the z-value specifies the distance above or below the surface elevation. These values must be expressed in the units of the coordinate system being used by the master grid. The default x-, y-coordinate pair corresponds to the geographic centre of the master grid. The default z-value is the lowest value in the master grid. You can modify these settings by entering a new value in the edit dialog box or by moving the slider bar to the left or right. When the slider bar is in use (indicated by the dashed line around it), the arrow keys can be used to modify this value.

The Surface and Lighting Tab

The Surface and Lighting tab contains settings that affect the visual characteristics of the rendered scene. Properties in this dialog box will affect all the grids listed in the Grid Layer Control.

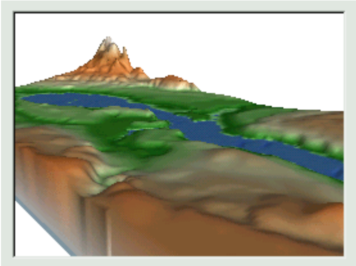


The **Lighting** section enables you to choose lighting settings implemented in a GridView scene. Each type of lighting is generated from a different source and each is user-controlled. Some light comes from a particular direction or position, and some light is scattered about the scene.

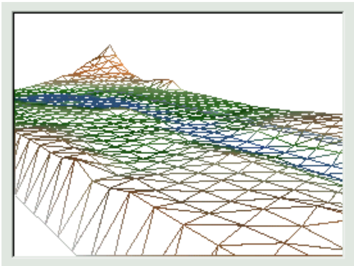
- **Specular** refers to light coming from a particular direction and tends to bounce off the surface in a preferred direction. A shiny surface such as metal has a high specular component, while a surface such as a carpet has almost none. Increasing the percentage of specular light results in strong shadow effects and more pronounced glare spots.
- **Diffuse** refers to light coming from a particular direction and is brighter if aimed directly down on a surface. When diffuse light hits the surface, however, it is scattered uniformly in all directions, so it appears equally bright no matter where the eye is located. Increasing diffuse light will intensify shadow effects.
  - **Ambient** refers to light coming from a source that has been scattered so evenly by the environment that its direction is impossible to determine. Increasing the ambient light will brighten the scene without casting shadows.
  - **Azimuth** is the angle in the x-, y- plane at which specular or diffuse light from the source position shines onto the grid. A value of 180 degrees indicates that the light is shining directly north in terms of the coordinate system of the grid file (i.e., directly along the x-axis of the coordinate system). This Azimuth setting operates in the same way as the viewing Azimuth setting illustrated in the next figure on the left.
  - **Inclination** is the angle at which the light shines down from the source position towards the midpoint of the surface, measured from the horizontal plane.

The **Modeling** section enables you to specify two ways of representing the rendered surface: Solid and Wireframe.

- **Solid** renders the scene with a continuous colour (see figure below on the left).
- ***Wireframe*** renders the scene with a wireframe mesh (see figure below on the right).



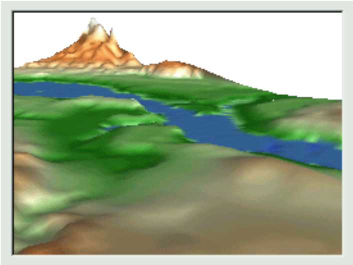
Solid displays a continuous colour surface.



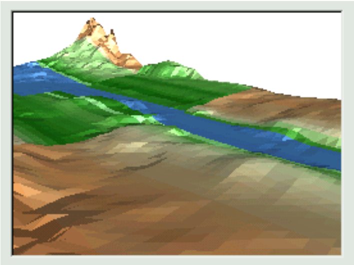
Wireframe displays a wireframe mesh.

The **Shading** section enables you to set the manner in which colours are applied to individual grid cells during the rendering process. There are two options: Smooth and Flat.

- **Smooth** renders a colour gradient between each grid cell when rendering a scene. This can greatly improve the appearance of low-resolution grids (see the figure below on the left).
- **Flat** assigns one colour to each grid cell during the rendering process (see the figure below on the right).



Smooth renders a continuous colour gradient.

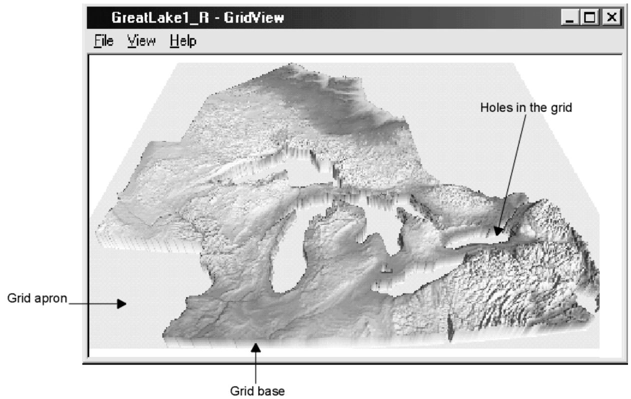


Flat assigns one colour to each cell.

The **Bitmap Size** section enables you to set the resolution of the rendered image. The Height and Width settings, measured in pixels, can increase or decrease the resolution or quality of the rendered image. High-resolution images can be panned but not zoomed in GridView. You can use MapInfo Professional or any raster/paint program to zoom images saved with the **Export to .bmp & .tab** command found on the File menu.

The **Scene** section enables you to set the characteristics that affect the visual appearance of the rendered scene. Increasing the shininess value of the surface will increase its overall gleam, while decreasing the shininess value will produce a duller, waxier looking surface. Specular Reflectance can be increased to produce more highlights on the surface.

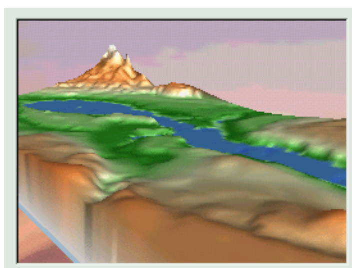
The **Null Value Colour** button enables you to set the colour applied to areas of a grid where there are null values. A null value is an arbitrary numeric value given to those grid cells that did not obtain a calculated value at the time of interpolation or that have been trimmed away at a later time. This colour can be generally seen in three different locations: the grid base, the grid apron, and the inside holes within the grid area (see the next figure). When this setting is chosen, a standard Windows Colour dialog box opens.



The areas affected by the Null Value Colour setting are easily identified.

The **Backdrop Colour** button enables you to set the colour applied to the backdrop or background of the rendered scene. When this setting is chosen a standard Windows Colour dialog box appears.

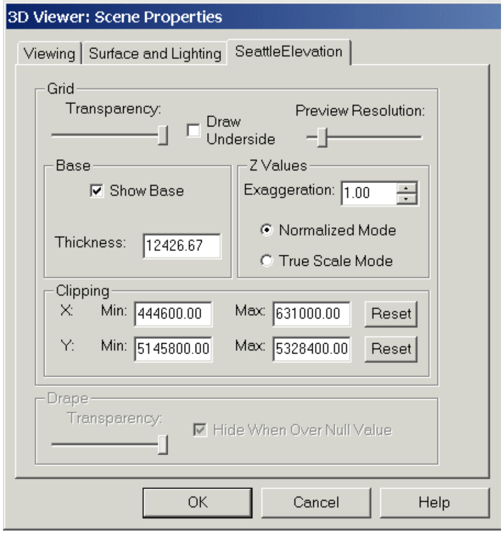
The **Backdrop Image** button enables you to set the .bmp image that is applied to the background of the scene (see the next figure).



An example of a scene using a backdrop image.

The Loaded Grids Tab

The Loaded Grids tab contains settings that directly affect the display of the highlighted grid file or drape file listed in the Grid Layer Control. The name of the highlighted grid is displayed as the title on the Loaded Grids tab (SeattleElevation in the next dialog box).

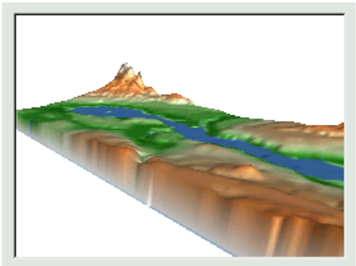


The Grid section enables you to set the display characteristics of the highlighted grid file.

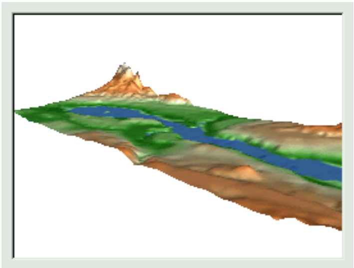
- The **Transparency** slider bar enables you to set the degree of transparency the highlighted grid will have. With the slider bar set fully to the right, the grid will display normally. As the slider bar is moved to the left, the grid will begin to vanish into the background. This can be particularly effective when you want to view two grids in the same scene, both with varying amounts of transparency. When the slider bar is selected (indicated by the dashed line around it), the arrow keys can be used to modify the degree of transparency.
- The **Draw Underside** check box enables you to render the underside of the highlighted grid. This is useful when the viewing location is set below the surface of the grid and the viewer is looking up from under the grid surface. When this option is set, the scene will render more slowly.
- The **Preview Resolution** slider bar enables you to set the degree of resolution for the preview window. When the slider bar is selected (indicated by the dashed line around it), the arrow keys are used to modify the degree of resolution. Although grids are rendered faster at lower resolutions, they have far less detail. With some grids, the resolution can be increased to a degree that proves to be extremely inefficient. If this occurs a warning message appears giving you the opportunity to abort the setting modification.

The **Base** section contains settings to control the display and size of the base of the grid.

- The ***Show Base*** check box specifies whether the base will be displayed or not (see the next figuresa). This setting is typically used in conjunction with the **Draw Underside** check box.

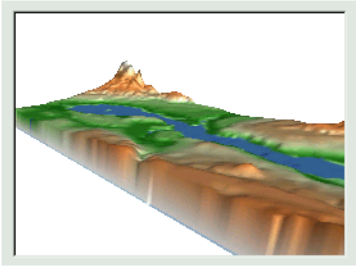


A rendered scene with Base

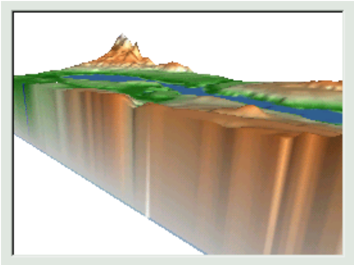


A rendered scene without Base

- The **Thickness** box enables you to set the thickness of the base in the coordinate units of the master grid (see the next figures).



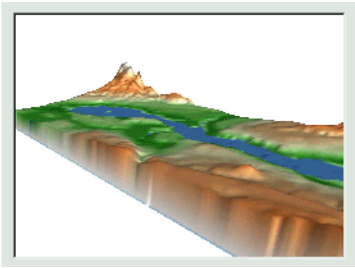
A rendered scene with the default base thickness



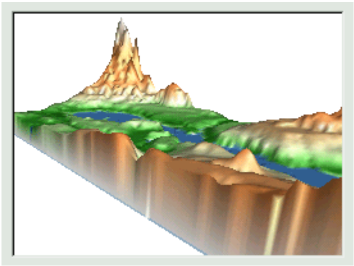
A rendered scene with an increased base thickness

The **Z Values** section enables you to control the vertical exaggeration of the highlighted grid in the Grid Layer Control.

- The **Exaggeration** box controls the degree of exaggeration of the z-value. This enables you to vertically stretch the rendered scene (see the next two figures).



A rendered scene with an exaggeration of two



A rendered scene with an exaggeration of five

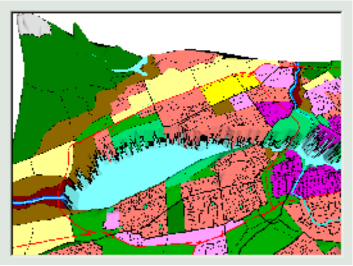
- The **Normalized Mode** option is generally chosen when the z-values are not in the same units as the grid coordinate system. For example, a grid displaying average family income will have coordinate units of either metres or degrees, depending on the grid projection, but will have z-units in dollars. The default exaggeration uses a 2:1 aspect ratio of the grid width (x-range) to grid height (z-range). The exaggeration value entered is multiplied by the arbitrary aspect ratio, allowing dimensionless vertical exaggeration control.
- The **True Scale Mode** option is chosen if the x-, y- coordinate system (geographic) units are the same as the z-value units. With a vertical exaggeration of one, the surface will be portrayed realistically.

The **Clipping** section describes the geographic area in the x-, y-coordinates of the highlighted grid that will be rendered in the scene. This is generally used when you want to render a small portion of a larger grid. The easiest way to specify these coordinates is to create a rectangular region in the Map window that describes the area to be rendered. Double-click on this region to display the Rectangle Object dialog box and the x-, y- bounding coordinates of the rectangle. Copy (Ctrl+C) and paste (Ctrl+V) these values from this dialog box to the GridView Scene Properties dialog box. You should note that the coordinates specified in the Rectangle Object dialog box contain commas, which are not accepted in GridView.

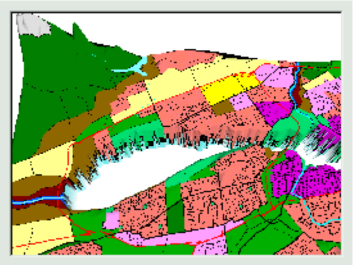
The **Drape** section contains settings that control the appearance of the selected drape file in the Layer/Grid Height list box. For more information on how to create and load drapes into a scene, see **Making 3D Drape Files on page 201**.

- The **Transparency** slider bar enables you to set the degree of transparency of the highlighted drape. With the slider bar set fully to the right, the grid will display normally. As the slider bar is moved to the left, the grid will begin to vanish into the background. When the slider bar is selected (indicated by the dashed line around it), the arrow keys can be used to modify the degree of transparency.

- The ***Hide When Over Null Value*** check box controls the display of a drape where it is geographically coincident with null values on the grid that it is draped upon (see the next two figures). These areas are made invisible .



This figure shows a drape file overlying a grid that has null values in the lake area.

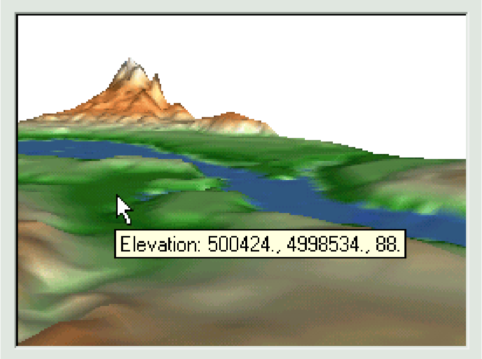


This figure shows how the Hide When Over Null Value setting behaves when checked.

The Preview Window

Located to the right of the 3D Viewer: Scene Properties dialog box, this Preview window displays how the rendered scene will look with the current settings. The window will automatically refresh when you modify the scene properties. The delay between modifying a setting and the subsequent refresh of the window can be altered in the GridView Preferences dialog box. For more information, see **Setting your Preferences on page 204**.

Another feature of the Preview window is the ability to query grid values. If you click while the cursor is inside the window, then a tool tip will appear with information from the grids in the Layer/Grid Height list box (see the next figure). This information includes the name of the grid(s) being queried, the x- and y- coordinates of the selected location, and the value at that location. This feature is available for any rendered scene.

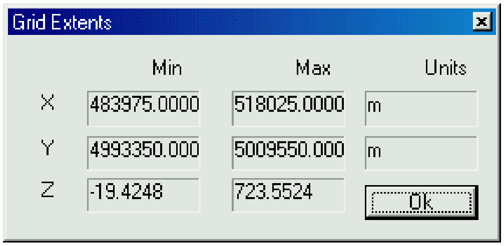


Click the left mouse button to get information on the grids in the Layer/Grid Height list.

The **OK** button in the 3D Viewer: Scene Properties dialog box enables you to close the dialog box and render the GridView scene with the current settings.

The **Cancel** button enables you to close the dialog box without rendering the GridView scene.

The **Grid Extents** button opens the Grid Extents dialog box which displays the x- and y- extents as well as a range of values of the highlighted grid file. The x- and y- values are expressed in the coordinate system units of the grid, and the z-value range is expressed in data units.



The **Unload** button enables you to remove the highlighted grid or drape file from the Layer/Grid Height list box.

The **Load** button enables you to add additional grids or drape files to the Layer/Grid Height list box.

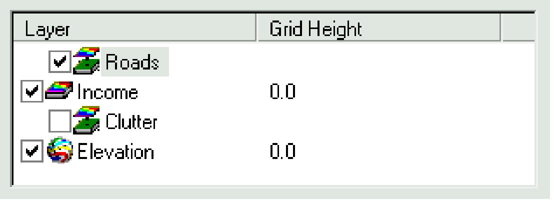
The Grid Layer Control

Grid Layer Control contains a listing of all the grids and drape files loaded into the memory of GridView.

- Master Grid is the first grid to be loaded into the Grid Layer Control. Any X, Y location or distance setting will be specified in the coordinate units of the Master Grid. In any single GridView session there can only be one Master Grid which is symbolized by a special icon. Once the Master Grid has been specified, it cannot be changed.
- A **Visibility** check box precedes each of the listed files whether they are grids or drapes. When checked, the associated grid/drape file will then be rendered in the scene.
- Additional Grids may be added to the Grid Layer Control by selecting the **Load** button on the bottom right of the Scene Preferences dialog box. These grids are symbolized by a different icon than the Master Grid.
- Drape Files may also be added to the Grid Layer Control by selecting the **Load** button on the bottom right of the Scene Preferences dialog box. When loading a drape, the selected drape file will be applied to the highlighted grid in the Grid Layer Control. If this drape file does not coincide with the highlighted grid a warning message will appear. More than one drape file can be loaded for any one grid; however, only one drape per grid can be visible at one time.
- Grid Height refers to the vertical distance the grid will be drawn at above the minimum value of the Master Grid. This distance value is specified in the grid coordinates of the Master Grid. All grids listed in the Grid Layer Control will be displayed in the order specified by the entered Grid Height value. The number of decimal places displayed in this setting can be modified in the GridView Preferences dialog box.

Working with Grid Layers

The Layer/Grid Height list in the Scene Properties dialog box displays the grids and drape files that are loaded into the memory of GridView.



The master grid is the first grid loaded into the Layer/Grid Height list box. Any x- or y- location or distance setting is specified in the coordinate units of the master grid. In any single GridView session, only one grid can be used as the master grid. The master grid is identified by the icon to the left of the grid name. Once the master grid has been specified, it cannot be changed.

The **Visibility** check box enables you to choose whether the grids or drapes will be rendered in the scene.

The **Load** button enables you to add additional grids or drape files to the list box. When loading a drape file, the selected drape file will be applied to the highlighted grid in the list box. If this drape file does not coincide with the highlighted grid, a warning message appears. More than one drape file can be loaded for any one grid. However, only one drape per grid can be visible at a time.

The **Grid Height** column of the **Layer/Grid Height** list box refers to the vertical distance the grid will be drawn at above the minimum value of the master grid. This distance value is specified in the grid coordinates of the master grid. All grids listed in the list box will be displayed in the order specified by the entered Grid Height value. The number of decimal places displayed in this setting can be modified in the GridView Preferences dialog box.

The GridView Menus

Once you render a scene, you can access the **GridView** menus.

The File Menu

The **Open** command enables you to add more grids to the rendered scene. It is similar to the Load button in the 3D Viewer: Scene Properties dialog box. Only grids can be opened with this feature.

The **Close All** command closes all currently open grid and drape files.

The **Export to .bmp & .tab** command creates a Windows bitmap of the rendered scene and a MapInfo .tab file so that the image can be viewed in MapInfo Professional. The resolution of the bitmap corresponds to the settings in the Bitmap Size section on the Surface and Lighting tab. For the best results make sure the Height or Width setting (whichever is smallest) in the Bitmap Size section is set to at least 2 000 pixels.

The **Open/Save Workspace** command opens an existing workspace or saves the current rendered scene to a workspace. When you save a workspace, all the settings for recreating the rendered scene are saved to an ASCII text file with a .gvw extension.

The **Print/Print Set Up** command controls how the rendered scene is printed and enables you to select an installed printer.

The View Menu

The **Scene Properties** command opens the Scene Properties dialog box so you can make further modifications. You can also open this dialog box by right-clicking anywhere within the rendered GridView scene.

The **Preferences** command opens the 3D Viewer: Application Properties dialog box where you can set the following values:

- The **Refresh Time** box enables you to set the length of the delay (in milliseconds) between the time a setting is changed in the Scene Properties dialog box and the time when the Preview window is refreshed.
- The **Preference Precision** box enables you to specify the number of decimals that will be displayed in the Scene Properties dialog box.
- The **Update While Rendering** check box when this option is enabled, the scene is drawn to the screen gradually as it is being rendered.

The Help Menu

You can access GridView help by clicking the **Help Topics** command from the **Help** menu.

Making 3D Drape Files

The process of draping involves combining georeferenced map objects with gridded data to create 3D perspective views of the area defined by the grid. Typically, the gridded information represents elevation; however, it can represent numeric data. In GridView, acceptable georeferenced map entities include symbols, lines, regions, text and other raster images such as air photos. Essentially any map attribute that can be displayed in a MapInfo Map window can be included in a drape. If you want to drape other raster images, you should orthorectify the images in the same projection system as the grid they will be draped on.

GridView drape files are created in MapInfo Professional with the help of Vertical Mapper. Three requirements must be met before you can create a drape:

- Ensure proper registration in GridView; the grid file upon which the drape will be placed must be present as one of the layers in the current MapInfo Map window. This layer does not have to be visible.
- All of the MapInfo layers contained in the drape file must be in the current Map window and be coincident with the geographic extents of the grid being draped. Only those map entities that fall within the extents of the grid will be included in the drape.

- The grid on which the drape is to be placed must highlighted in the Grid Manager.

Note The process of creating a drape is similar to the Save Window As command from the File menu in MapInfo Professional. In Vertical Mapper, however, you can specify the resolution with the Create Drape File command.

1. In the Grid Manager, choose a grid from the list of open grids.
2. Click the **3D View** button and choose the **Make 3D Drape File** command.

   This process creates two files: a bitmap (.bmp) of the Map window and a drape file (.drp) that contains the registration information needed by GridView.
3. Choose a grid from the **Layer/Grid Height** list in the Scene Properties dialog box.
4. Click the **Load** button and choose **Drape Files (*.drp)** from the **Files of Type** list.
5. Choose the desired drape file.
6. Click the **Open** button.

Exploring the Drape File Resolution Dialog Box

The Drape File Resolution dialog box contains the name of the grid file upon which the drape will be created and the number of pixels of the drape file.

By default, the file name for the grid file is based on the name of the grid highlighted in the Grid Manager, and the drape file is rasterized to 1 000 pixels in width. You may enter any value in the edit box. Remember that large bitmap files can quickly consume hundreds of megabytes of disk space and require significant memory to process efficiently. If you intend to use the drape file to create promotional materials such as large scale maps or posters, a resolution of 2 000 pixels or more is recommended.

Note The maximum width possible depends on system resources available. The greater the width, the higher the resolution of the drape file and the more system resources required to create it.

13

Other Commands

This chapter covers how to set preferences for grid file management and dialog box usage and how to exit from Vertical Mapper.

In this chapter:

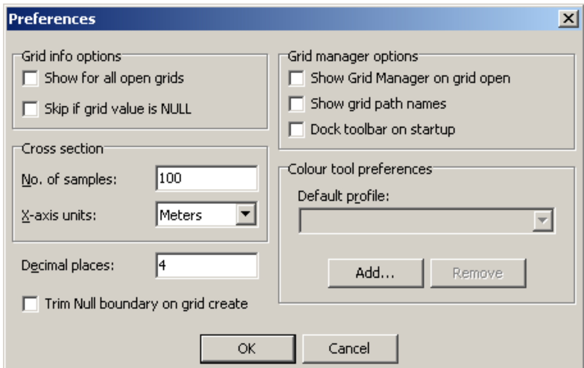
Setting your Preferences

You can determine the default settings that control grid file management and dialog box usage in the Preferences dialog box.

- From the Vertical Mapper menu, choose the **Preferences** command.

Exploring the Preferences Dialog Box

The Preferences dialog box enables you to assign a number of default settings.



The **Show for all open grids** check box—enables you to show all grids in the Grid Info dialog box regardless of whether the grid file is active or inactive.

The **Skip if grid value is NULL** check box—enables you to hide grid information in the Grid Info dialog box if the selected cell has a null value.

The **Show Grid Manager on Grid Open** check box—enables you to set the Grid Manager so that it appears every time a grid is opened using the Open command from the File menu in MapInfo Professional.

The **Show grid path names** check box—enables you to show the full directory path for the grid file in the Grid Manager.

The **Dock Toolbar on Startup** check box—enables you to dock the Vertical Mapper toolbar to the top of the MapInfo Professional window.

The **No. of Sample** check box—enables you to set the number of points (samples) taken along the line of cross section that is used to create the x-y plot of distance versus grid value in the graph window; the greater the number of samples, the more detail will be displayed in the cross-section plot.

The **X-Axis Units** list—enables you to choose distance units for the x-axis of cross-section graphs.

The **Colourizer Preferences** section—enables you to set a custom default colour profile for all numeric grids created.

- The **Profile** list enables you to select a custom default colour profile that will be applied to all grid files generated during a Vertical Mapper session.

- The **Add** and **Remove** buttons enable you to add any number of profiles to or remove them from the list. In the absence of any preferred colour profile, the Vertical Mapper default gradient colour pattern is used. This profile cannot be edited.

The **Decimal Places** box—enables you to set the number of decimal places of accuracy that will be visible when working with entered or returned values. You can enter values that contain more than the set number of decimal places and use them in the operation. However, the display of these values will always be rounded off to the Decimal Places setting.

The **Trim Null boundary on grid create** check box—enables you to select the trimmed grids without any Null Boundary. When the Trim Null Boundary is selected the Null Boundary is removed from the resultant grid during the trimming process.

The **OK** button applies preference settings to all subsequent Vertical Mapper sessions until you make modifications to your preferences.

Exiting Vertical Mapper

You exit Vertical Mapper in the same way you exit MapInfo Professional.

- From the Vertical Mapper menu, choose the **Exit Vertical Mapper** command.

H

I

K