

User Guide

ISOMAP 3D Surface Modelling

geo&soft[®]
international

Geo Soft di ing. G. Scioldo

Summary

Chapter 1 - Introduction to the ISOMAP family	1
Introduction to the ISOMAP family	1
Chapter 2 - System requirements and program installation	3
System requirements	3
Program installation.....	3
Chapter 3 - The program protection	8
Program registration.....	8
Chapter 4 - How to update the program	9
Procedure for updating the program	9
Autoupdating requirements and troubleshooting	10
Chapter 5 - User Interface	11
Usage Notations.....	11
User Interface and Data Entering.....	11
User Interface: Menu Bar and Menus	11
The Input Dialogue Windows	12
Data Input With Tables.....	13
Message Windows	14
Help On Line	14
Chapter 6 - Open a new project	15
Project Menu	15
New Project Command	15
Open Project Command.....	16
Delete Project Command.....	16
Printer Setup Command.....	17
Chapter 7 - Grid Generation	18
Operation Grid Generation	18
Weighted Average.....	18
Kriging	18
Method of the Weighted Average of Polynomial Surfaces	19
Procedure.....	19
Edit Menu - Grid Generation	19
Data Points Command.....	20
The "Data Points" Dialogue box.....	20
Gridding Parameters Command	20
The "Gridding Parameters" Dialogue Box.....	20
Example	21
Hidden Mesh Selection Command	22
Calculate Menu - Grid Generation	23
Execute Calculation Command.....	24
Z(X,Y) Calculation Command	24
Calculation of the volume Command	24
Print Menu	24

Parameters Command.....	24
The “Graphical Parameters” Dialogue Box.....	24
The <Manual Colour Setting> Button	27
Print Grid Command	28
The Graphical Output Preview Window.....	28
Print Vertical Section Command.....	29
The “Vertical Section Configuration” Dialogue box.....	29
Export to SLK format Command.....	30
Configure Command.....	30
Exit Menu	30

Chapter 8 - Slope Map **31**

Slope Map (ISOMAP module).....	31
Procedure.....	31
Edit menu - Slope Map.....	31
Hidden Mesh Selection Command	32
Calculate Menu - Slope Map.....	32
Execute Calculations Command.....	32
Z(X,Y) Calculation Command	32
Calculation of the volume Command	32
Print Menu - Slope Map	32

Chapter 9 - Exposure Map **33**

Exposure Map (ISOMAP module).....	33
Procedure.....	33
Edit menu - Exposure Map.....	33
Hidden Mesh Selection Command	33
Calculate Menu - Exposure Map.....	34
Execute Calculations Command.....	34
Z(X,Y) Calculation Command	34
Calculation of the volume Command	34
Print Menu - Exposure Map	34

Chapter 10 - Grid Difference **35**

Grid Difference (ISOMAP module).....	35
Procedure.....	35
Edit menu - Grid Difference.....	35
Hidden Mesh Selection Command	35
Select Subtrahend Grid Command.....	36
The “Select Subtrahend Grid” Dialogue Box	36
Calculate Menu - Grid Difference.....	36
Execute Calculations Command.....	36
Z(X,Y) Calculation Command	36
Calculation of the volume Command	36
Print Menu - Grid Difference	36

Chapter 11 - Linear Transformation **37**

Linear Transformation (ISOMAP module).....	37
Procedure.....	37
Edit Menu - Linear Transformations.....	37
Hidden Mesh Selection Command	38
Linear Transformation Parameters Command.....	38
The “Linear Transformation Parameters” Dialogue box	38
Calculate Menu - Linear Transformation.....	38
Execute Calculations Command.....	38
Z(X,Y) Calculation Command	38
Calculation of the volume Command	39
Print Menu - Linear Transformation	39

Chapter 12 - Filtering **40**

Filtering (ISOMAP module)	40
Procedure.....	40
Edit Menu - Filtering	40
Hidden Mesh Selection Command	41
Select Filter Command.....	41
The "Select Filter" Dialogue box	41
Edit Filter Command	41
Calculate Menu - Filtering	41
Execute Calculations Command.....	41
Z(X,Y) Calculation Command	41
Calculation of the volume Command	42
Print Menu - Filtering.....	42
Chapter 13 - Grid Duplication	43
Grid Duplication (ISOMAP module)	43
Procedure.....	43
Chapter 14 - DTM Import	44
DTM Import (ISOMAP module)	44
Procedure.....	44
Chapter 15 - VID Import	45
Import VID File (ISOMAP module)	45
Procedure.....	45
Chapter 16 - XYZ Import	46
Import XYZ file (ISOMAP module)	46
Procedure.....	46
Chapter 17 - DXF Import	47
Import DXF file (ISOMAP module)	47
Procedure.....	49
Chapter 18 - ASC Import	50
Import ASC file (ISOMAP module)	50
Procedure.....	50

Chapter 1 - Introduction to the ISOMAP family

Introduction to the ISOMAP family

The ISOMAP family is an integrated software package that allows one to create a digital terrain model (DTM) that can be used for further elaborations, such as rockfall analysis and groundwater modelling.

ISOMAP is the module that is used to calculate and render surfaces through contour lines or coloured and shaded areas with a high degree of precision. The program allows the representation of the surface both in topographic map and perspective view forms.

The calculation is performed in two stages. The first stage consists of creating a regular grid from a collection of arbitrarily positioned points. The second stage consists of drawing the surface using the previously created grid. The first stage can be performed using three methods: the inverse distance method, the kriging method, and a new interpolation and extrapolation method based on a weighted average of polynomial surfaces.

The use of this polynomial algorithm makes it possible to generate points that are external to the area of the sampled locations in such a way as to maintain the trend of the surface, even when this surface cannot be approximated by a simple horizontal plane.

This feature is especially useful for those problems that deal with the surface gradient, such as flow line tracing or rockfall analysis. The commonly used extrapolation algorithms based on the weighted average (including kriging) can lead to large errors, for example to the inversion of the flow direction in the peripheral areas of the map.

The use of a polynomial algorithm to create a grid also yields a more realistic surface of the area where the sampled locations can be found, as the evaluation is less sensitive to the spatial distribution and point density.

The regular square grid is then interpolated with bi-cubic splines to obtain a continuous surface that is continuously differentiable and which passes through the grid nodes.

The ISOMAP module has the following basic operations:

- **Grid Generation**
- **Slope Map**
- **Exposure Map**
- **Grid Difference**
- **Linear Transformation**
- **Filtering**
- **Grid Duplication**
- **DTM Import**
- **DXF Import**
- **VID Import**
- **ASC Import**

These operations are described later in this manual.

The **ROTOMAP** module is dedicated to 3D rockfall analysis, and allows the complete design of rockfall protective systems. It features true 3D modelling and many other options for model calibration and barrier design.

The "Introduction to the ROTOMAP module" and the subsequent chapters are not included in the present version of the manual.

The **INQIMAP** module is dedicated to groundwater modelling. It leads to calculations from those based on simple analytical solutions to those that incorporate advanced and complex numerical techniques.

The "Introduction to the INQIMAP module" and the subsequent chapters are not included in the present version of the manual.

The ISOMAP family is a comprehensive package: it can directly produce high-quality graphic outputs, or export them to DXF or EMF formats, which preserve the vectorial quality of the printouts, even when the files are imported into external editors such as Microsoft Word or Corel Draw.

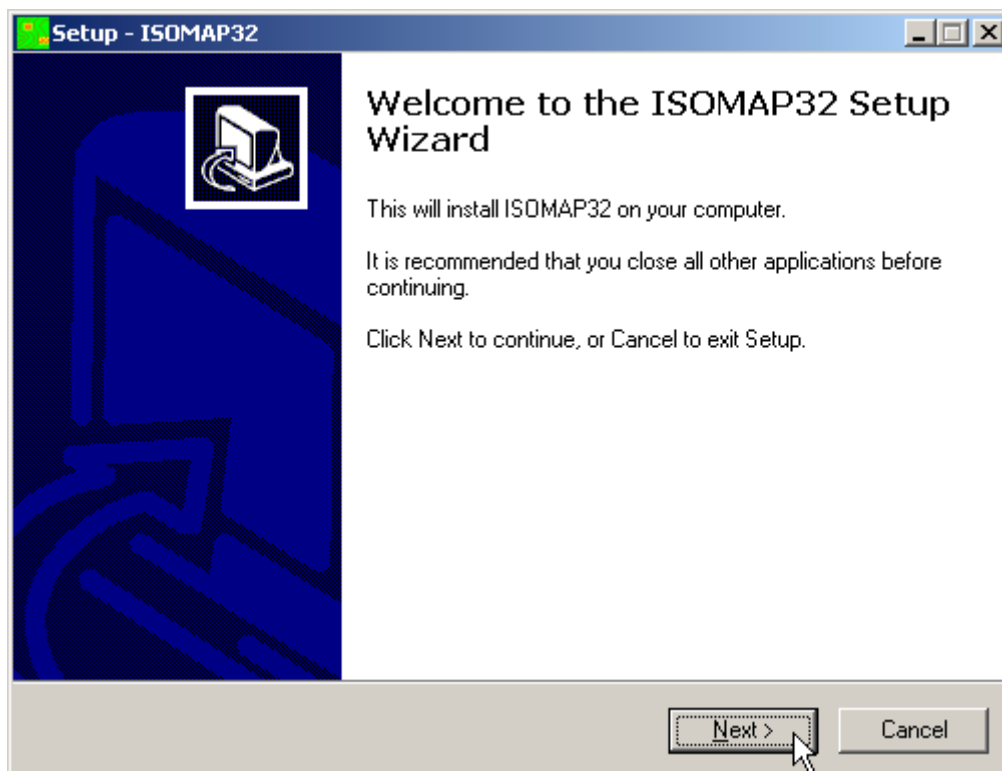
Chapter 2 - System requirements and program installation

System requirements

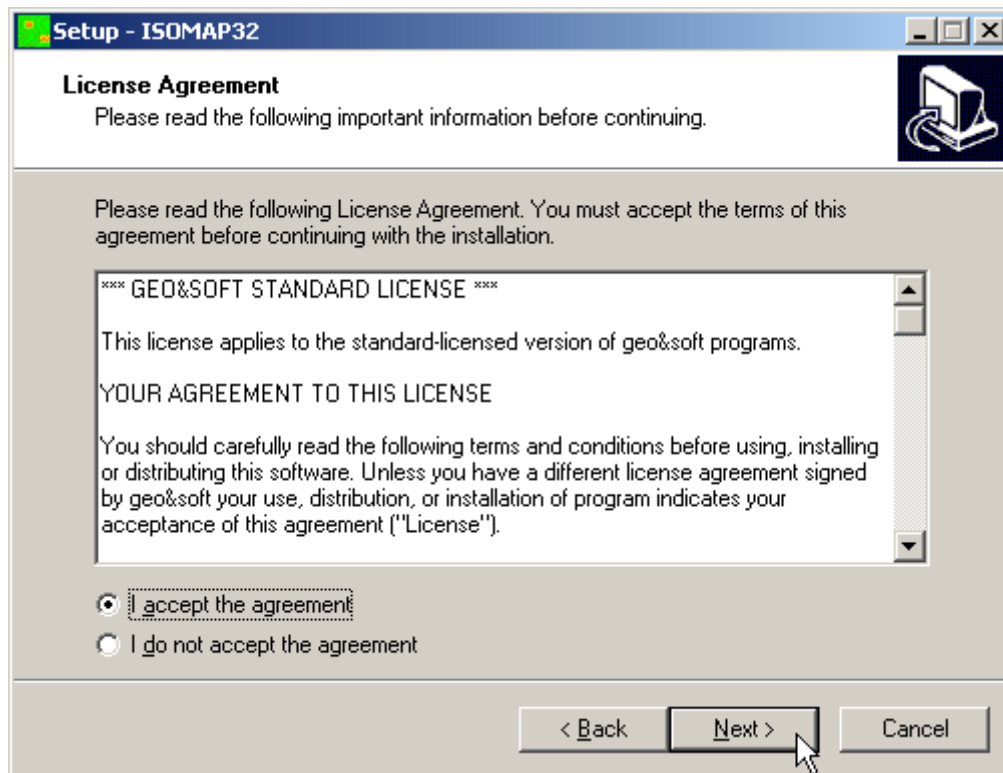
- Pentium® class processor
- Microsoft® Windows® 95 OSR 2.0, Windows 98, Windows Me, Windows NT®* 4.0 with Service Pack 5 or 6, Windows 2000, or Windows XP
- 64 MB of RAM (128 MB recommended)
- 100 MB of available hard-disk space
- CD-ROM drive
- A printer driver **must** be installed, even if the printer itself is not connected to the PC.

Program installation

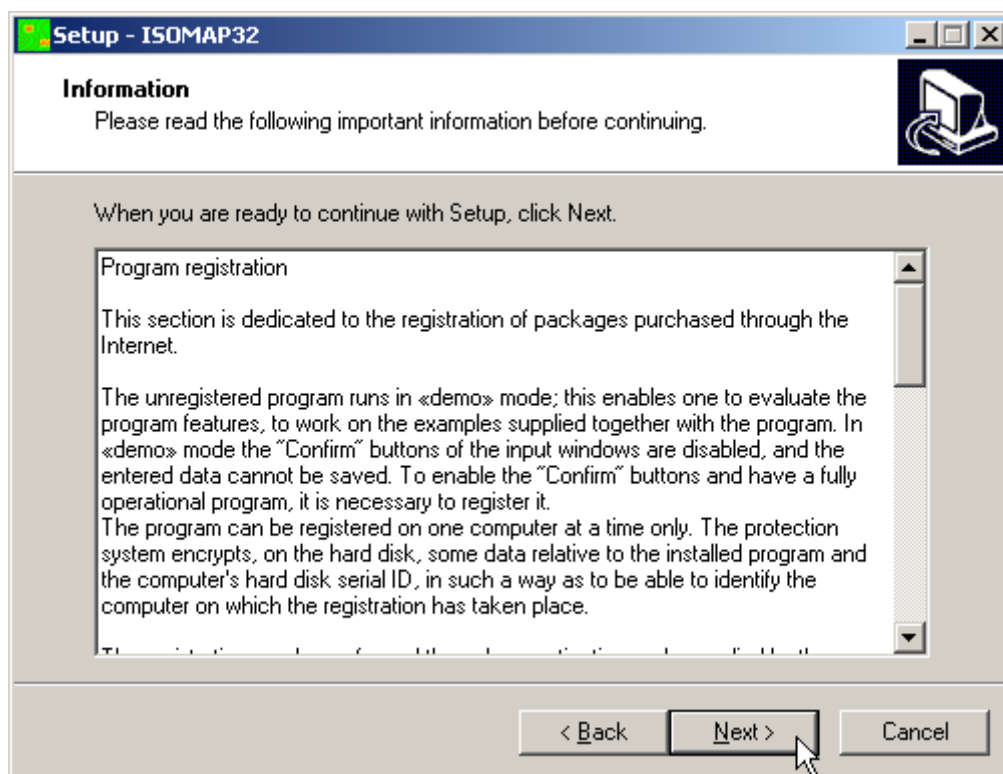
To install the ISOMAP program, run **ISOMAP32SETUP.EXE** from the CD-ROM or from the folder where you downloaded and saved the setup program.



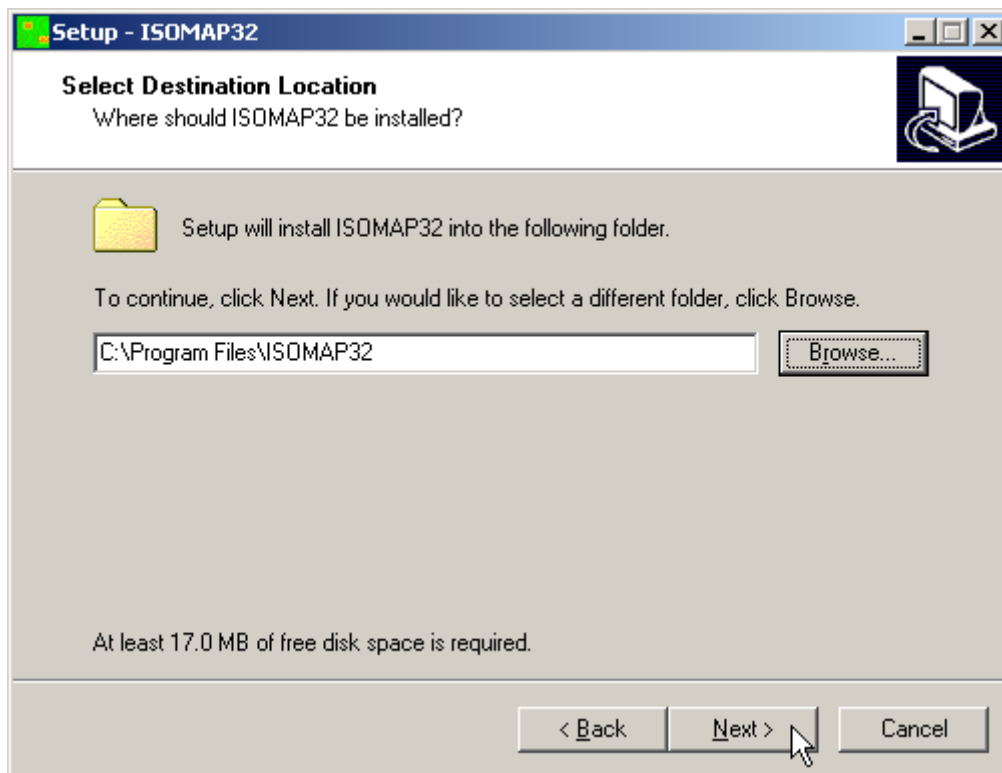
Press the **NEXT** button to start the installation.



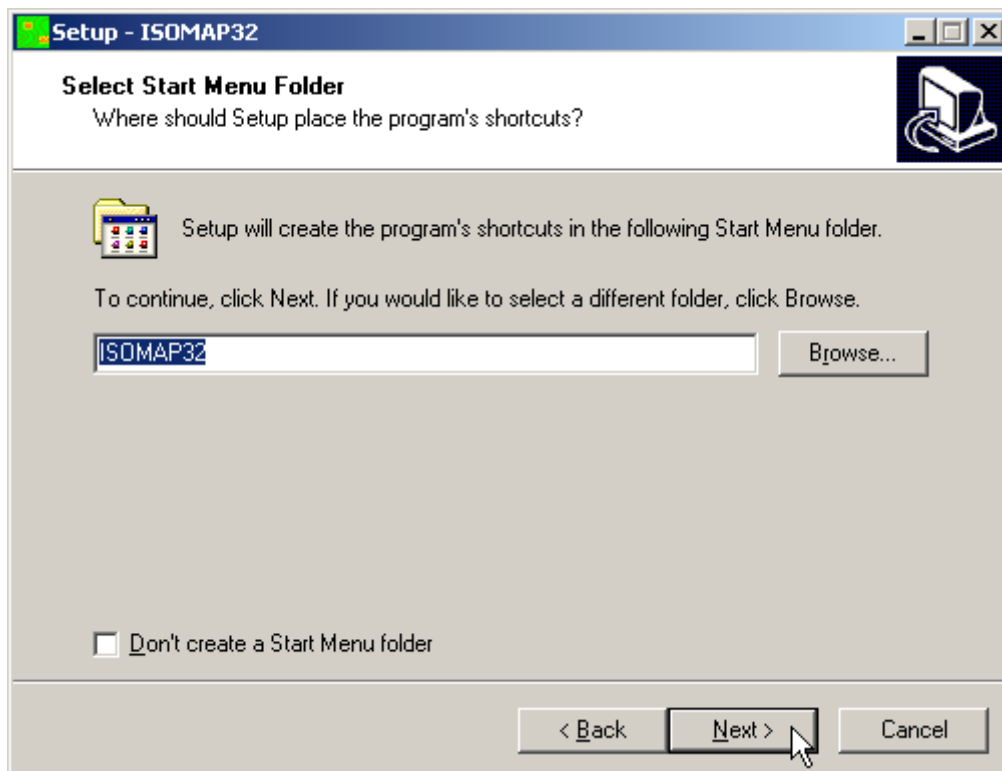
Carefully read the License Agreement and press the **NEXT** button.



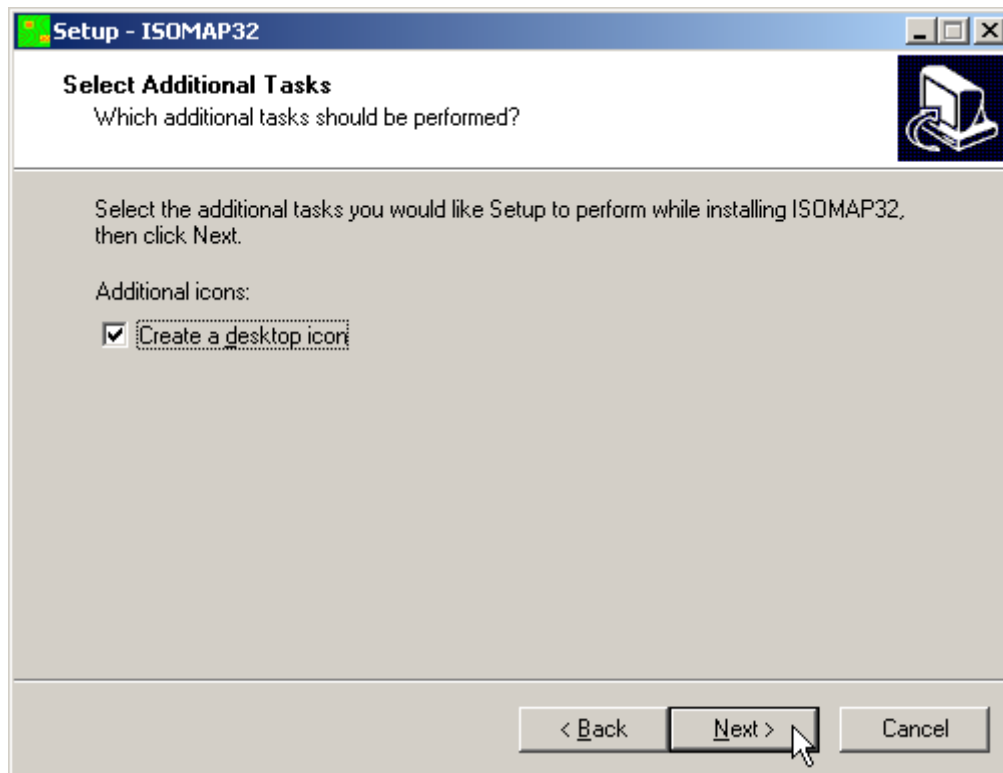
Carefully read the informations about the registration of the software and press the **NEXT** button.



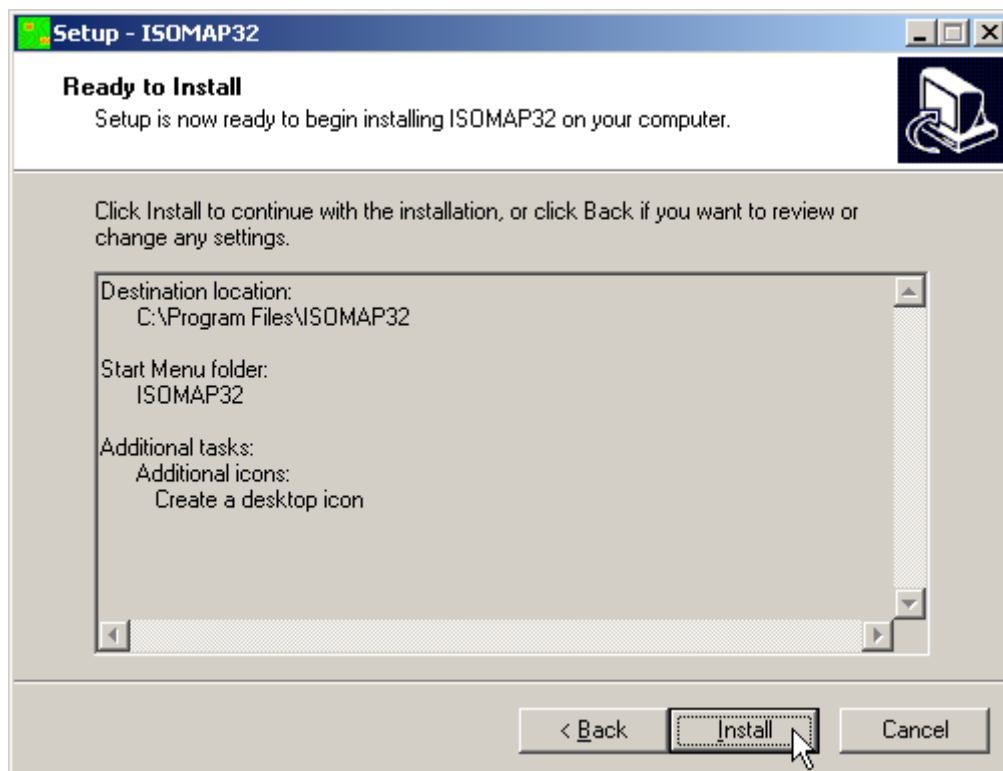
Select the folder where the **ISOMAP32** directory would be created and press the **NEXT** button.



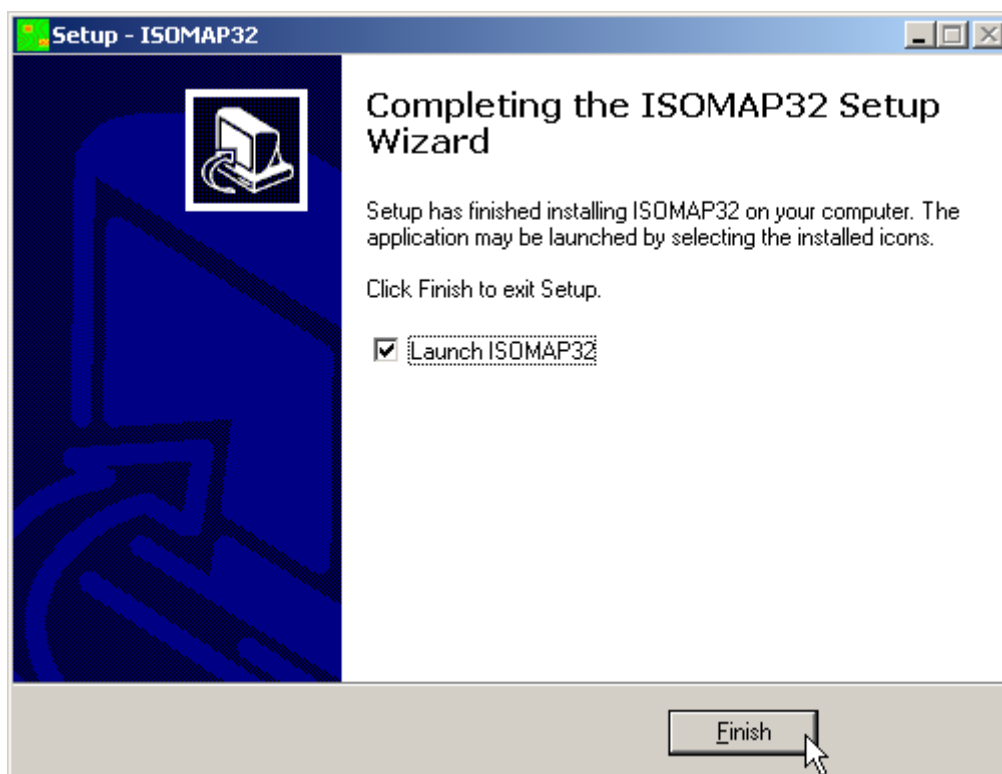
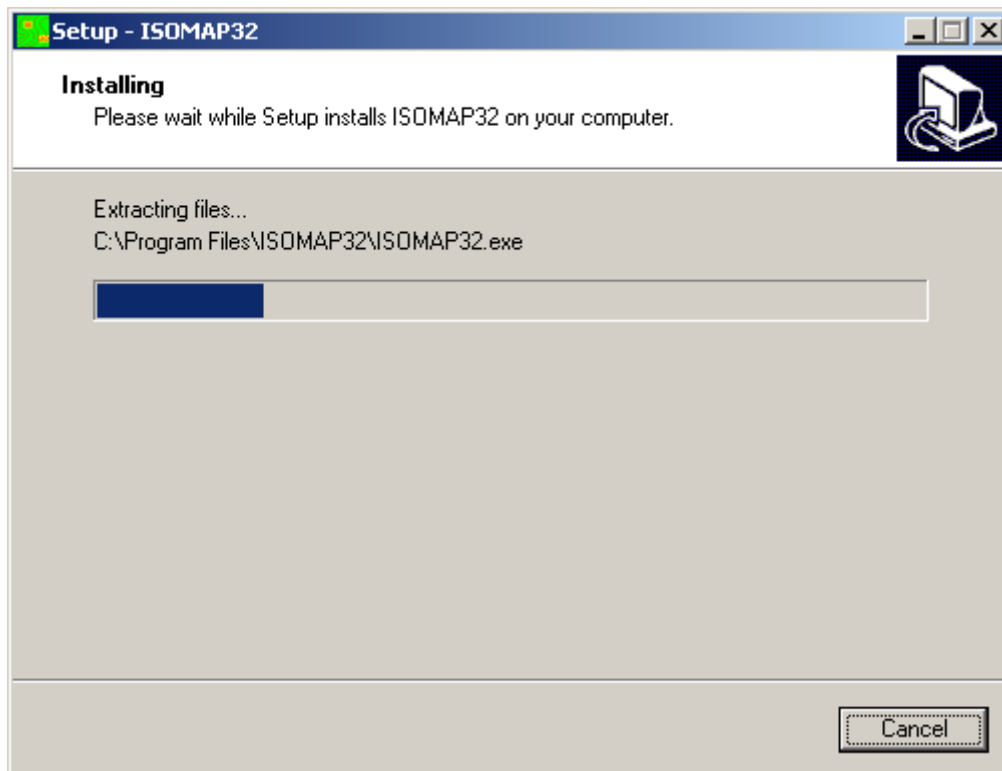
Press the **NEXT** button to create the **ISOMAP32** group in the Start menu.



Choose if you want an icon to be created on your desktop, or not.



Press the **INSTALL** button to continue the installation procedure ...



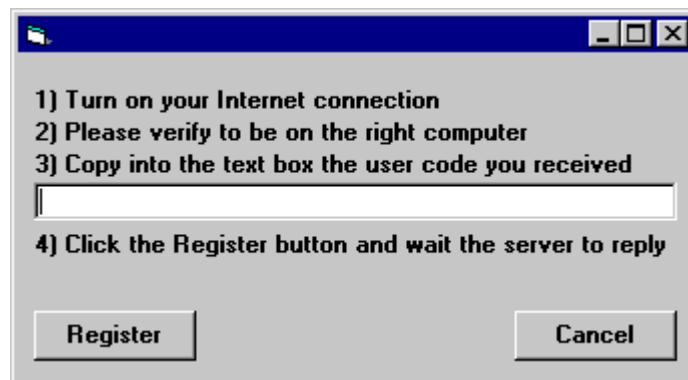
Once completed the installation, press the **END** button to exit the setup procedure and to start **ISOMAP32**.

ISOMAP32 can be run by clicking the **Start** button, then selecting the **ISOMAP32** group in the **Programs** menu, and clicking **ISOMAP32**.

Chapter 3 - The program protection

Program registration

This section is dedicated to the registration of packages purchased through the Internet.



The unregistered program runs in «demo» mode; this enables one to evaluate the program features, working on the examples supplied together with the program. In «demo» mode the «Confirm» buttons of the input windows are disabled, and the entered data cannot be saved. To enable the «Confirm» buttons, and have a fully operational program, it is necessary to register it.

The program can be registered on one computer only at a time. The protection system encrypts, on the hard disk, some data relative to the installed program and to the computer hard disk serial ID, in such a way as to be able to identify the computer on which the registration has taken place.

The registration can be performed, by the manufacturer or supplier, with an **activation code**.

We would like to stress that the encrypted protection data on the hard disk are bound in a unique way to the hard disk itself: subsequently, one can no longer use the activation code to install the program on other computers.

Let us describe the procedure for the registration of the software:

1. select the **Registration** command from the Help menu.
2. turn on your Internet connection
3. copy into the box the activation code you received
4. click the <Registration> button and wait the server to replay

Chapter 4 - How to update the program

Procedure for updating the program

We believe that the best advertising is a client who shows to the colleagues a state-of-the-art product, so we have always made available the free updates of our packages.

Now a new service is available for free: **AutoUpdater** automatically searches and downloads the new components from our web site **www.geoandsoft.com**, saving the older versions in a backup folder.

If you prefer, you still can download the updated full setup from our web site, of course.

To download the updated components with AutoUpdater, follow this procedure:



1. Start the program
2. Click the "**Update**" button at the bottom of the first window.
3. Wait for the program to verify if some updated component is available on our servers. If the program finds new components, it shows the list: each component has a check box which allows to choose if a single component has to be downloaded, or not. AutoUpdater automatically checks the boxes of the components whose download is suggested, and leaves unchecked the files which could have been modified by the user, like the colour configuration files.
4. Select the files you want to download and click the "**Update**" button.
5. Once the files have been installed, AutoUpdater runs the updated program.

AutoUpdater can also be configured to automatically start at each run of the program. Simply check the “update at startup” check box, and AutoUpdater will automatically search for new components each time the program is run.



Autoupdating requirements and troubleshooting

The autoupdating procedure requires an active Internet connection.

The search for new components is directly performed by the program, while the download is performed by AutoUpdater: for this reason, both the main program and AutoUpdater need the right to access to the Internet.

All the autoupdating procedures use the HTTP protocol, port 80, and automatically recognize if a proxy server is used on the local network to access the Internet.

If you experience some problems with the autoupdating procedure, they may be due to a FIREWALL: ask your System Administrator to give the right to access the Internet to the executable of the program (*32.exe) and to the AutoUpdater.exe module, in the program folder. No “act as a server” right is needed.

If particularly hard security rules disallow giving those access rights, remember you can download the updated full setup, from our web site, through your web browser.

Chapter 5 - User Interface

Usage Notations

Some typographical notations and keyboard formats are used in this manual to help locate and interpret information more easily.

Bold print is used to indicate command names and related options. Characters appearing in bold print should be typed exactly as printed, including spaces.

Words written in *italics* indicate a request for information.

CAPITAL letters are used to indicate computer, printer, directory, and file names.

User Interface and Data Entering

The user interface is designed to be easy to use and powerful and is supported by complete on-line help. This help contains practical hints and the theoretical background, where applicable. It should reduce the requirement of frequently consulting the printed manuals.

All the commands are located inside a menu bar. Each menu contains a list of commands that one can select with the mouse or the keyboard. The arrangement of the menus, designed with ergonomic criteria, follows the logical order of the operations, inhibiting the access to further operations until all the necessary data have been entered.

The interface layout is maintained in all of our programs, to simplify, as much as possible, the transition from one program to another to avoid having to learn different commands and procedures for similar functions (such as entering data or managing files).

Let us examine the general components that are available in the user interface of geo&soft programs.

User Interface: Menu Bar and Menus

The Menu Bar manages the access to all the program commands. The goal of the menu design is to offer an ergonomic, simple, and understandable arrangement of the commands.

The menus used to perform a complete operation are normally ordered left to right and top to bottom. When possible, the following scheme is used: definition of the name of the project, entering the required data, performing the calculation, and generating the output as a preview or final print.



The programs automatically disable some menus or commands when the execution of such menus or

commands is not yet feasible. Typical examples are: a calculation menu is disabled when not all the data needed for the calculation have been completely entered, or a print menu is disabled if not all the calculations have been executed.

To choose a command from a menu, use the following procedures:

- ☞ Point the menu that is to be activated, click the left button of the mouse, then point the command to be activated, and click the left button of the mouse again. If one does not wish to select any command from the menu that is open, exit the area of the menu before releasing the mouse button.

The Input Dialogue Windows

The different menu commands can perform an immediate action, or display a dialogue window in order to input or edit the various data sets.

When a dialogue window is visible, all the actions that do not pertain to it are ignored. Hence, it is necessary to close the dialogue window to resume the normal use of the program.

GRIDDING PARAMETERS

Calculation algorithm Weighted Average of Polynomial Surfaces

Order of the polynomial surface [-] 0

Mesh size [m]	125
Threshold distance [m]	5550
Weighting exponent [-]	4
Minimum X coordinate [m]	350
Maximum X coordinate [m]	3225
Minimum Y coordinate [m]	230
Maximum Y coordinate [m]	355
Range of the X coordinates	350 < X < 3125
Range of the Y coordinates	230 < Y < 256
Range of the Z coordinates	200 < Z < 214.5

OK Cancel Help

The calculation of the grid can be performed using the Kriging or the Weighted Average of Polynomial Surfaces; a zero-order polynomial surface is a special case that corresponds to the ordinary inverse-distance weighted average method.

Some fundamental tools are used inside the dialogue windows: data fields such as text and list boxes, and buttons.

The text boxes are used to input numerical values and text strings. Most of the editing keys (HOME, END, INS, DEL, etc.) can be used inside these fields. If one sees a small arrow pointing down on the right side of the data field, one has a list box.

To scroll the list box and select an item, do as follows:

- ☞ click the arrow, and then click the item to be selected.

There are three buttons in the dialogue window:

- <Ok> - this button saves the entered information and goes on to the next phase.
- <Cancel> - this button closes the Dialogue window without saving the just entered data, or without executing the command.
- <Help> - this opens a window that contains general information on how to use the dialogue windows.

To use the buttons:

- ☞ click the button

To edit the data inside the dialogue window, use the following keys:

- TAB - moves the cursor to the next field; when the cursor is inside the last visible field the cursor goes to the first control button of the Dialogue window. To return to the previous field, press SHIFT+TAB.

- ENTER - moves the cursor to the next field. If the cursor is positioned on one of the window buttons, the corresponding command is executed.
- BACKSPACE - cancels the last character that has been entered.
- DEL - cancels the character to the right of the cursor.
- ESC - closes the Dialogue window without saving the entered values or without executing the command. The <Cancel> button will do the same.
- UP ARROW/ DOWN ARROW - these are used in multiple fields, or those fields that have a list box.
- LEFT ARROW/ RIGHT ARROW - moves the cursor to the previous or next field.
- HOME -moves the cursor to the beginning of the field.
- END - moves the cursor to the end of the field.

A yellow box with a short text that explains the meaning of the value to be entered, can be seen in the lower part of the window. If the text is not completely visible, click the yellow box to read the complete text.

Data Input With Tables

Tables are used to enter long sequences of numerical values and/or text strings. The keys to be used are the following:

Point	X Coord. [m]	Y Coord. [m]	Elevation [m]	Name
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				
15				

The planimetric coordinates should be entered (in metres) from an arbitrary origin.

OK Cancel Insert Delete Help

- TAB - moves the cursor to the first button in the window. When one presses this key again, the cursor is moved to the next button.
- SHIFT+TAB - moves the cursor to the previous button.
- ENTER - moves the cursor into the next input field. If the cursor is positioned on one of the window buttons, the corresponding command is executed.
- PAGE UP - moves the cursor up 15 lines.
- PAGE DOWN - moves the cursor forward 15 lines.
- UP ARROW - moves the cursor to the input field directly above.
- DOWN ARROW - moves the cursor to the input field directly below.
- LEFT ARROW / RIGHT ARROW - moves the cursor to the field respectively to the left or to the right of the

current position.

- HOME - moves the cursor to the beginning of the line.
- END - moves the cursor to the end of the line.
- F2 - copies the field contents to the extended editing field under the title bar in order to facilitate the editing of the long strings. When working in this editing field, please **REMEMBER** to press ENTER, even before clicking the <Ok> button. Double-clicking a field has the same effect as pressing the F2 key.

The tables have two additional buttons:

- <Insert> - creates an empty line before the one in which the cursor is positioned.
- <Delete> - deletes the line in which the cursor is positioned.

IMPORTANT SUGGESTION: you can copy data to or from other programs such as Microsoft Excel:

The data entered in the table can be copied in order to be pasted into another table.

To copy the table's contents:

 press the key combination CTRL+C. The contents will be copied into the Clipboard of Windows.

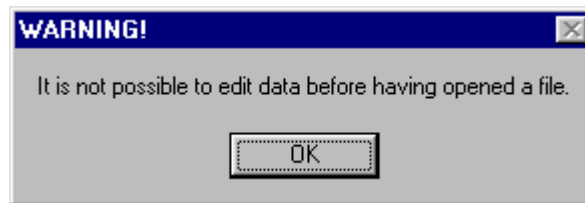
To paste the Clipboard contents into the table:

 press the key combination SHIFT+INS, or CTRL+V.

A yellow box with a short text that explains the meaning of the value to be entered, can be seen in the lower part of the window. If the text is not completely visible, click the yellow box to read the complete text.

Message Windows

The function of these windows is to give information to the user concerning the system status, as in the case of an error due to an improper use of the program.



Help On Line

A complete Help On Line is available. It is possible to ask for information or suggestions related to the commands or the use of the program. In order to access the help on line, proceed as follows:

1. Position the cursor on a field of a dialogue window and press F1.
2. Use the Summary from the Help menu.

Chapter 6 - Open a new project

Project Menu

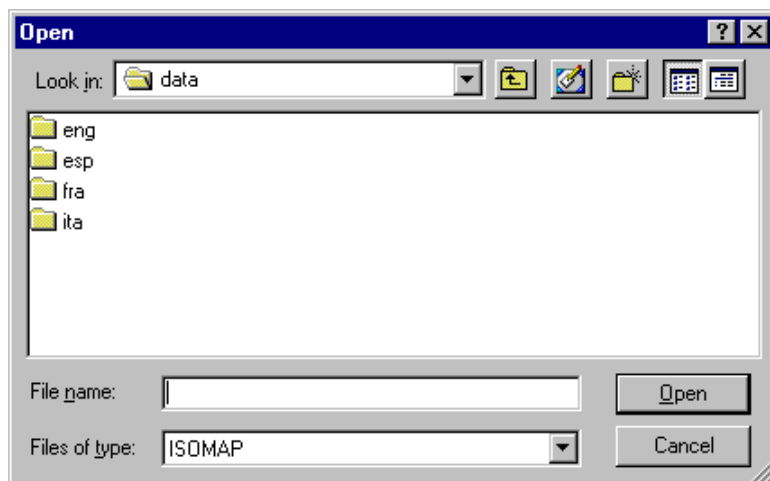
This menu contains all the commands that are used to open and delete the files which contain all the project data. As long as the project name has not been defined, the other menu items are disabled.



The last opened projects are shown in the lower part of this menu.

New Project Command

In order to operate with this program, it is necessary to open a project; this can be a new project or an already existing one. To open a new file:



- choose the New Project command from the menu by first clicking the Project menu item and then on the New Project command. Type the name of the new file in the File Name field and confirm.

The name of the file in use will be reported in the title bar of the main window.

The name of the file should not contain spaces and/or punctuation marks. It is not necessary to indicate any extension, since the extension “.NFJ” is automatically added: for example, given the name “TEST01”, the file name “TEST01.NFJ” will be internally used by the program.

When opening a new project, it is necessary to choose which operation has to be performed. Only one

operation can be associated to each project name, and it cannot be changed later on.

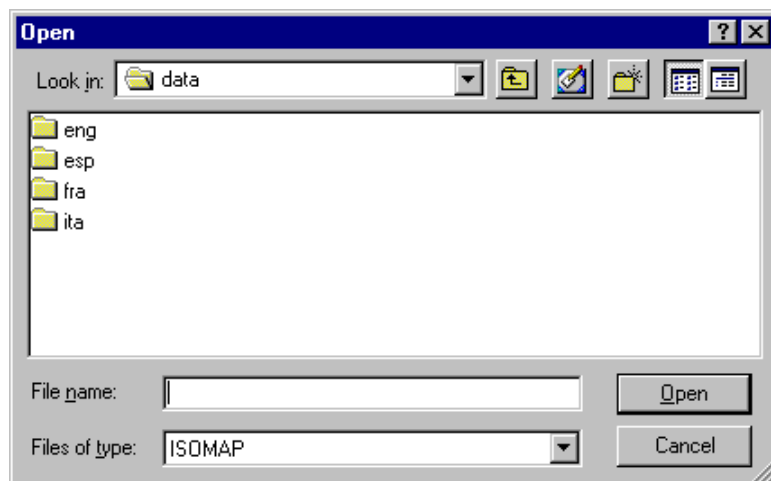
It is possible to create a series of projects, each of which uses the results of a previously elaborated project as a “starting point”: this will be the **Source Project**.



There is no specific command in the program to save data, as these are automatically saved after each data entry or elaboration.

Open Project Command

In order to operate with this program, it is necessary to open a project; this can be a new project or an already existing one. To open an existing file:



☞ choose the Open Project command from the menu by first clicking the Project menu item and then on the Open Project command from the menu. Type or double click the name of the new file in the File Name field and confirm.

The name of the file in use will be reported in the title bar of the main window.

There is no specific command in the program to save data, as these are automatically saved after each data entry or elaboration.

Delete Project Command

This makes it possible to delete all the files pertaining to a project. Before using this command, one should be sure that these data are no longer necessary. Particular care must be taken if the project has been used as a Source Project for other elaborations. Before deleting the files, the program asks for confirmation.

Printer Setup Command

When selecting this command, one calls the printer configuration dialogue window. Here one can verify and modify the default printer setup or select another printer from the list of the installed ones.

Chapter 7 - Grid Generation

Operation Grid Generation

The basis of any project is the generation of a regular grid. The square grid is generated using one of the available methods, starting from a set of arbitrarily positioned points sampled on the surface under examination. The more regular the disposition and density of the input points are, the more reliable the final result will be, regardless of which calculation method is used. Let us consider the calculation methods that can be used.

Weighted Average

The value for unsampled locations is equal to the weighted average of the values of the nearby samples. The weighted average takes the following form:

$$Z'(p) = \frac{\sum_{i=1}^n (Z(i) * (1 / D(i, p)^\alpha))}{\sum_{i=1}^n (1 / D(i, p)^\alpha)}$$

where:

$Z'(p)$ = estimated value at point p

$Z(i)$ = sampled value at point i

$D(i,p)$ = distance between point p and point i

alpha = weighting exponent

n = number of sampled points

This kind of interpolation is unbiased if the sum of the weights equals one. This is true for the inverse-distance weighted method because the sum of the weights divided by the sum of the weights is equal to one.

The exponent on the distance function above can be altered. Altering the exponent on the distance affects the relative weights of the points. In all cases, a sample that is further from the point to be estimated will receive a lower weight.

As the exponent is increased, the relative influence of more distant points decreases.

A problem that exists with this method is that you have to make a guess at what the exponent should be, and there is no assurance that your guess is correct.

Kriging

The Kriging algorithm, also called “best linear unbiased estimator”, was developed for mining geology. It is “best” because it minimizes the error variance in the estimate, “unbiased” because the weights sum to one, and “linear” because it is a simple weighted average. It also uses a weighted average method to calculate the value at unsampled locations, but rather than guessing at the relationship between similarity of values and distance (like we do when we guess at the exponent in inverse-distance methods), the relationship is calculated from the data using the semivariogram.

Once the lognormal semivariogram has been calculated, the weights $L(i)$ can be obtained to estimate the value of an unsampled location. The Kriging takes the following form:

$$Z'(p) = \sum_{i=1}^n (Z(i) * L(i))$$

where:

$Z'(p)$ = estimated value at point p

$Z(i)$ = sampled value at point i

n = number of sampled points

This set of weights has the property $\sum L(i) = 1$

and guarantee the minimisation of the general expression that represents the variance of the error associated to the estimate relative to point p .

$$2 * \sum_{i=1}^n (L(i) * G(D(i, p)) - g(D(p, p)) - \sum_{i=1}^n (\sum_{j=1}^n (L(i) * L(j) * G(D(i, j))))$$

$$2 * \sum_{i=1}^n (L(i) * G(D(i, p)) - g(D(p, p)) - \sum_{i=1}^n (\sum_{j=1}^n (L(i) * L(j) * G(D(i, j))))$$

where:

$G(D(x,y))$ = average variance associated to the distance between the points x and y

n = number of sampled points

This minimisation is obtained through the solution of a system of $n+1$ linear equations.

Method of the Weighted Average of Polynomial Surfaces

One of the greatest problems of the previously described methods is the impossibility of estimating values outside the range defined by the maximum and minimum values of the sampled points. This problem has the immediate consequence of preventing one from extrapolating, in a significant way, the values beyond the area that is actually covered by the sampled points.

Hydrogeology or rockfall analysis applications, based on the treatment of partial derivatives of the surface (gradients), are in fact extremely sensitive to irregularities, even small ones, of the surface.

For this reason a more complex algorithm has been made available, that is based on an auxiliary polynomial estimator $Z''(i)$ which represents the regional trend of the variable under examination.

The estimate therefore assumes the following form:

$$Z'(p) = \sum_{i=1}^n (Z''(i) * (1/D(i, p)^\alpha)) / \sum_{i=1}^n (1/D(i, p)^\alpha)$$

where:

$Z'(p)$ = estimated value at point p

$Z''(i)$ = estimated value in function of the value assumed at point i

$D(i,p)$ = distance between point p and point i

alpha = weight exponent

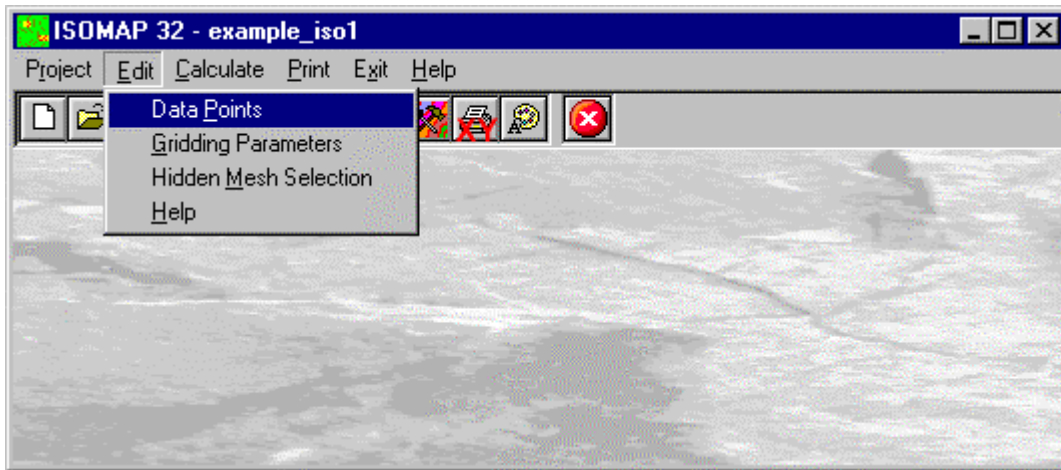
n = number of sampled points

Procedure

- Select the "New Project" command from the "Project" menu
- Choose a new name for the project
- Select "Grid Generation" in the "Operation" window
- Use the "Edit Points" in the "Edit" menu to input the spot sampled points
- Use the "Grid Parameters" in the "Edit" menu to choose how the grid will be calculated
- Select "Calculation" on the menu bar

Edit Menu - Grid Generation

This menu is used to enter and edit the input data used in the gridding operations.



Data Points Command

This enables one to enter and edit the input data points.

The "Data Points" Dialogue box

The following data should be entered in the "Data Points" window:

Point	X Coord. [m]	Y Coord. [m]	Elevation [m]	Name
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				
11				
12				
13				
14				
15				

The planimetric coordinates should be entered (in metres) from an arbitrary origin.

OK Cancel Insert Delete Help

- *X and Y Coordinates*: the planimetric coordinates should be entered (in metres) from an arbitrary origin.
- *Elevation*: the ground elevation (or another physical quantity) that has to be represented on the Z-axis.
- *Name*: this is an optional caption of the data points. There are two special characters that can be added at the beginning of the point names (notice that those characters will not be printed on the graphical output): when the point name begins with the @ character, the value will be used in the calculation but will not be printed; when the point name begins with the # character, the data point will be printed (if one of the *Sampled locations* options has been selected) but will not be used in the calculation.

Gridding Parameters Command

The Isomap module creates a regular grid that calculates the values of the physical quantity for each node of a square mesh using different interpolation algorithms. The required parameters are entered in the Gridding Parameters dialogue box.

The "Gridding Parameters" Dialogue Box

The following data are entered in the "Gridding Parameters":

GRIDDING PARAMETERS	
Calculation algorithm	Weighted Average of Polynomial Surfaces
Order of the polynomial surface [-]	0
Mesh size [m]	125
Threshold distance [m]	5550
Weighting exponent [-]	4
Minimum X coordinate [m]	350
Maximum X coordinate [m]	3225
Minimum Y coordinate [m]	230
Maximum Y coordinate [m]	355
Range of the X coordinates	350 < X < 3125
Range of the Y coordinates	230 < Y < 256
Range of the Z coordinates	200 < Z < 214.5

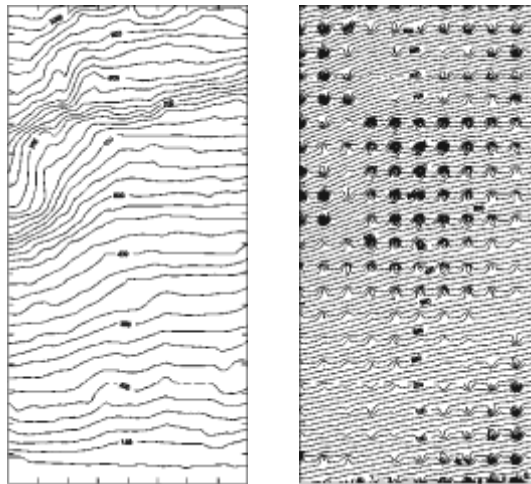
OK Cancel Help

The calculation of the grid can be performed using the Kriging or the Weighted Average of Polynomial Surfaces; a zero-order polynomial surface is a special case that corresponds to the ordinary inverse-distance weighted average method.

- **Calculation Algorithm:** the calculation of the grid can be performed using the Kriging or the Weighted Average of Polynomial Surfaces; a zero-order polynomial surface is a special case that corresponds to the ordinary inverse-distance weighted average method.
- **Order of the polynomial surface:** the basic concept of the Weighted Average of Polynomial Surfaces is that the best fitting polynomial surface is calculated, as a first step, from the data points, then a gradual vertical shift allows one to obtain a final surface that passes through the sampled locations. Going away from the sampled points, the calculated surface converges to the best fitting polynomial surface. A zero-order polynomial surface ($Z = a$) is a special case that corresponds to the classic inverse-distance weighted average method. A first-order polynomial surface ($Z = a + b X + c Y$) is a dipping plane that gives, for example, a good approximation for a simple water table only sampled in three wells. The surfaces of the second and third order are more complex and can better approximate a surface with many sampled locations. These surfaces require a homogeneous distribution of the sampled points because the gradient of the surface rapidly increases as it becomes distant from the sampled area.
- **Mesh size:** the mesh size (in metres) should be such as not to exceed 500 grid elements along each X and Y axis; if this value is exceeded, the mesh size is automatically adjusted.
- **Threshold distance:** this is the radius (in metres) of the circle wherein the points that will be considered to calculate a single grid value lie. As the used methods are based on an inverse-distance weighted average, it could be a good idea to keep this value sufficiently large so as to consider all the sampled points in the calculations, thus also obtaining a smoother surface with a small mesh size.
- **Weighting exponent:** this is the exponent that is applied to the inverse distance which is used in the weighted averages. The greater the exponent, the greater is the influence of a sampled point on the calculated grid. This means that too small exponent values can generate a surface with “peaks” around each sampled point, while too large exponent values can generate a “benched” surface. Values between 4 and 6 can normally yield good results.
- **Minimum X coordinate:** together with the ordinate of the lower side, this defines the lower left corner of the grid.
- **Maximum X coordinate:** together with the ordinate of the upper side, this defines the upper right corner of the grid.
- **Minimum Y coordinate:** together with the abscissa of the left side, this defines the lower left corner of the grid.
- **Maximum Y coordinate:** together with the abscissa of the right side, this defines the upper right corner of the grid.

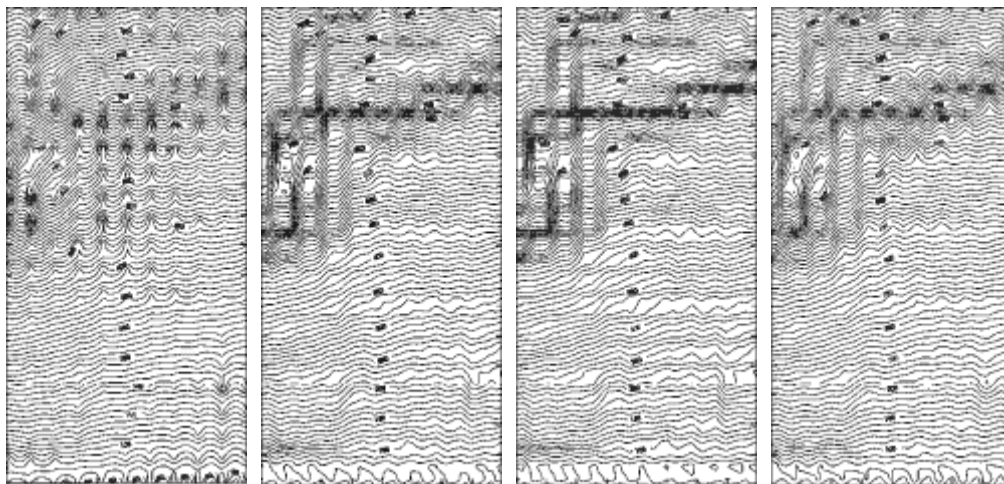
Example

Example: an original surface, with data points regularly sampled on each grid node, has been made denser to show the effect of the *Weighting exponent*. The best result has been obtained with Exponent = 3.



Original surface Exponent = 3

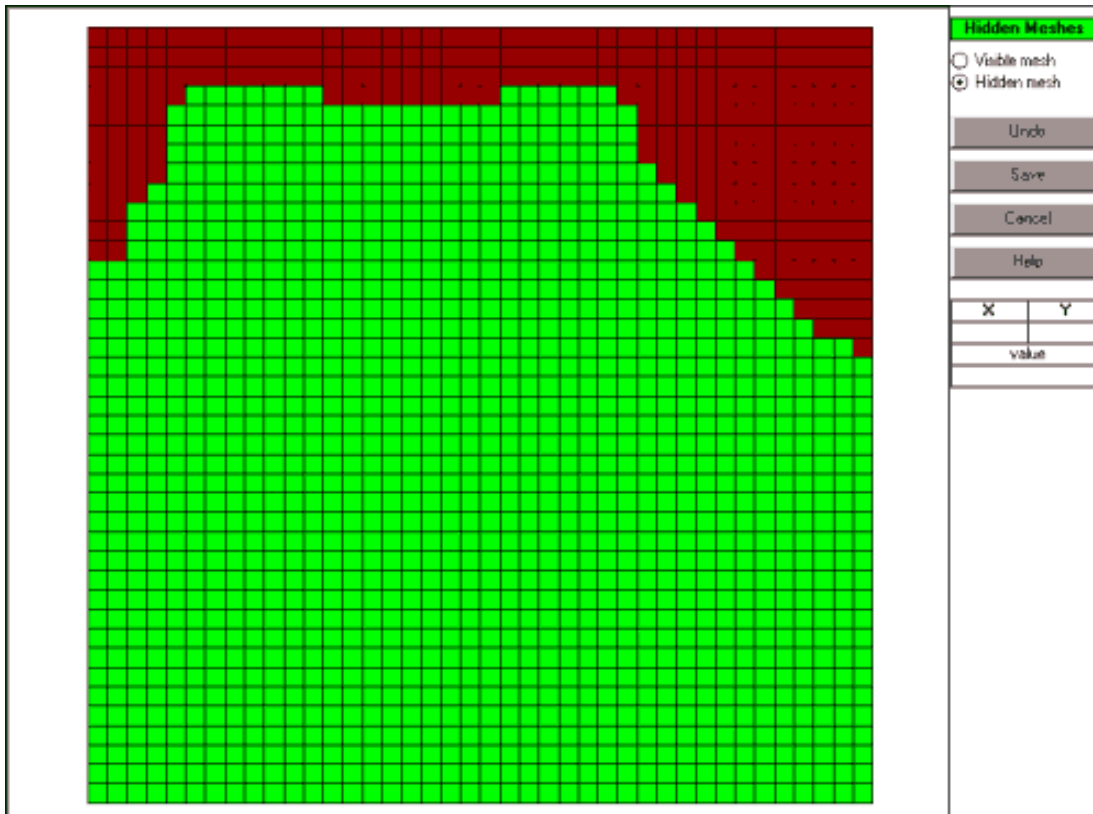
Exponent values 1 and 2 produced a surface with “peaks” around each sampled point (particularly emphasised with the minimum value); on the other hand, Exponent values 4 and 8 produced a “benched” surface (particularly emphasised with the maximum value).



Exponent = 1 Exponent = 2 Exponent = 4 Exponent = 8

Hidden Mesh Selection Command

This command allows one to hide some of the grid meshes. A graphical editing window is opened and the grid is shown: by default, all the meshes are not hidden, and appear as green squares.



The following procedure allows one to hide some meshes:

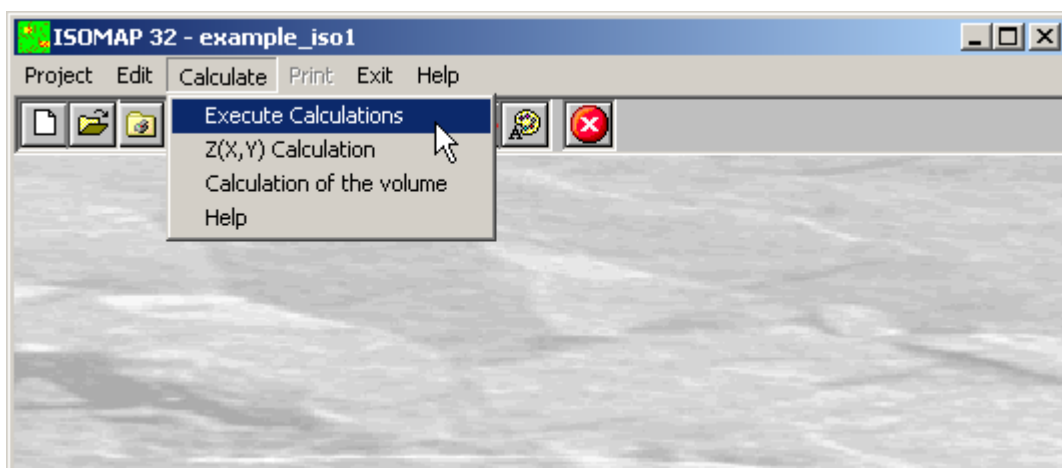
- <Hidden>/<Shown>: select one of the two option buttons to choose whether to hide or show the meshes. The hidden meshes will appear as red squares.

A single left-click assigns the currently selected status to a single grid mesh. Holding down the left button and moving the mouse over the grid allows a continuous (multiple) selection of the meshes. Finally, a right-click fills an area that is delimited by a line of previously selected meshes.

- <Cancel editing>: cancels the very last editing operation.
- <Save>: saves the selection of hidden meshes.
- <Cancel>: closes the window without saving any modifications.

Calculate Menu - Grid Generation

With this menu, one can calculate the regular grid using the entered data points and the previously defined parameters.

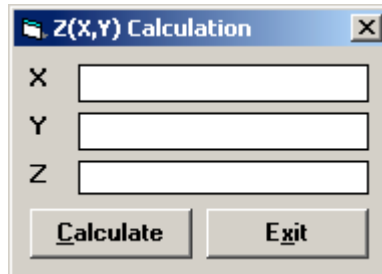


Execute Calculation Command

With this command, the program generates the grid. Use the print menu to view the results of the elaboration. For more information, see the Grid Generation Operation.

Z(X,Y) Calculation Command

With this command, one can evaluate the "z" value of any point of the surface; enter the coordinates of the point in the window and click the <Calculate> button.



The program calculates the required value using a bi-cubic splines interpolation algorithm, and displays the result in the "Z" field.

Calculation of the volume Command

With this command, the programme calculates the volume (surface integral) of the whole grid, and shows it in a message box.

Print Menu

The Print menu is used to configure the graphical output and plot the surface in Map, Wireframe View, or Solid View form.



Parameters Command

With this command, the "Graphical Parameters" dialogue window is shown.

The "Graphical Parameters" Dialogue Box

The "Graphical Parameters" dialogue window allows the configuration of most of the graphical parameters (see also the "Configure" item of the Print menu).

- *Map, Wireframe View, Solid View*: one can here choose the representation type.

The three buttons on this dialogue window allow one to select different parameter sets.

<Geometric Parameters> Button

- *Scale*: this is the scale factor that has to be used in the graphical output. It is the denominator in the fraction 1:scale
- *Labeled contour lines interval*: this is the elevation difference between two labeled contour lines. One ought to remember that the number of lines is inversely proportional to the given interval. A too small interval will slow down the drawing speed.
- *Unlabeled contour lines interval*: this is the elevation difference between adjacent unlabeled contour lines. One ought to remember that the number of lines is inversely proportional to the given interval. A too small interval will slow down the drawing speed.
- *Smooth factor*: this is the number of elements into which each grid element is divided, in order to represent more detailed contour lines; the larger this number the more accurate and attractive the drawing will be. If a high value is used there will consequently be a longer computing time and a larger file size, which increase proportionally with the square of the Smooth factor. If the Smooth factor is equal to one, a special faster algorithm is used. The Smooth factor value can vary between one and nine.
- *Minimum Z coordinate*: this defines the minimum threshold value of the contour lines that have to be represented. This parameter can be useful, for example, when plotting concentration contour lines where it would be erroneous to represent negative values. The range between minimum and maximum is an open interval.
- *Maximum Z coordinate* this defines the maximum threshold value of the contour lines that have to be represented. This parameter can be useful when one wants to eliminate values that are too high, such as the borders of polynomial extrapolations.
- *Zero range to hide*: this parameter allows one to hide the contour lines close to zero. This option is useful, for example, for the representation of magnetometric data.
- *Vertical exaggeration factor*: this factor enables one to stress the drawing, thus enhancing the anomalies of the plotted quantity.
- *Viewpoint distance*: this is only used in the *Solid View*. It is the absolute distance (in meters) between the viewpoint and the centre of the grid. This parameter is not used in the *Wireframe View*, because, in this case, the viewpoint distance is considered to be infinite.
- *Viewpoint Vector X, Y, Z components*: these define the direction of the viewpoint from the centre of the grid. The program disregards the vector magnitude, and automatically considers only the direction. It is used both in the *Solid View* and in the *Wireframe View*.
- *Light Vector X, Y, Z components*: these define the direction of the light point from the centre of the grid. If the three coordinates are equal to zero, no light effect is added. These parameters are used in the *Map* and

in the *Solid View*.

- *Sampled locations*: one can decide if and how the sampled locations will be drawn. The available options are: a simple cross, a cross labeled with the value of the sample, a cross labeled with the name of the location, or no representation at all.

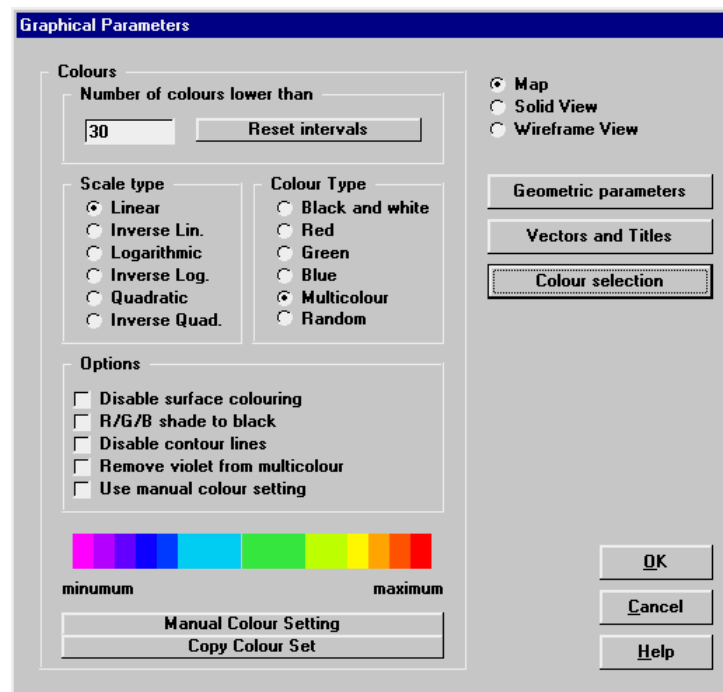
<Vectors and Titles> Button

- *Flow vectors*: with this option it is possible to draw flow vectors as small arrows positioned on the grid nodes. There are three options: Direction - unit vectors are drawn, oriented towards the dip direction; Velocity - the velocity vectors are drawn, as defined in Darcy's law, with the lengths proportional to the product of the hydraulic head gradient and the hydraulic conductivity; Gradient - flow vectors are shown as arrows oriented towards the dip direction: the vector length is proportional to the dip of the surface. No vector is drawn with the No Vector option.
- *Vector step*: this is used to space the vectors further apart; when assigning a value equal to 3, a vector, for example, will be drawn every three grid nodes.
- *Vector length / mesh width ratio*: the maximum (or unit) length of the vectors is, by default, equal to the mesh width. It is possible to alter this value by assigning a vector length / mesh width ratio that is different from one.
- *Lines*: this is used to draw a set of continuous flow lines. The required data are: the X,Y coordinates of the end points of an imaginary segment, and the number of equally spaced points along this segment through which the flow lines will pass.
- *Titles*: entering a title or sub-titles is not mandatory.
- *Legend Caption*: when a *Map* or a *Solid View* is coloured, a legend is automatically added. It is possible to add a caption to describe the plotted quantity.
- *Legend check box*: unchecking this box, the legend will not be plotted; this option is useful to export a "clean" graphic output to other programs like GIS.

<Colour Selection> Button

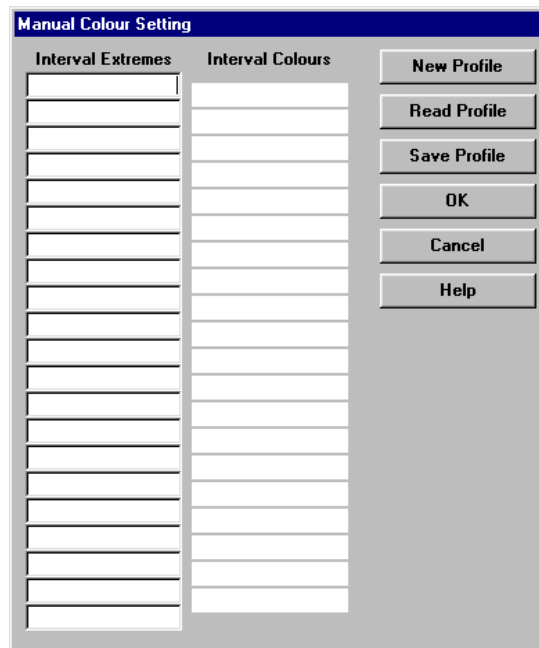
- *Number of colours lower than*: the program automatically detects the optimal contour line intervals, using multiples of $1.0E^n$, $2.0E^n$ and $5.0E^n$. Setting the *Number of colours lower than* parameter, the program is forced to choose a value of the contour line interval that produces a number of contour lines (and colour ranges) lower than the given value.
- *"Reset intervals" button*: this button activates a procedure that recalculates the contour line intervals as a function of the previous parameter.

- *Scale type*: the colour shade distribution can be organised using different scale types: linear, inverse linear, logarithmic, inverse logarithmic, quadratic or inverse quadratic.
- *Colour type*: the colour sets can be generated in different ways: Black and White, Red, Green, Blue, Multicolour, and Random. With the Black and White option one obtains a grey-scale output; With the Red, Green or Blue option the selected colour shades towards white; the Multicolour options allows one to create outputs using a continuous scale of colours, and the Random option assigns a colour that is randomly chosen to each level.
- *Disable surface colouring*: when this option is activated, the program does not colour the areas between the contour lines.
- *R/G/B shade to black*: with this option the selected colour will shade towards black instead of towards white.
- *Disable contour lines*: when this option is activated, the program no longer draws the contour lines; if both *surface colouring* and *contour lines* are disabled, nothing appears on the graphical output.
- *Remove violet from multicolour*: this option allows one to start the multicolour scale (if selected) from the blue instead of from the violet colour.
- *Use manual colour settings*: one can choose to manually define the contour line values and the colours to fill the area between the adjacent contour lines. To manually define those parameters, click the <Manual Colour Setting> button.
- *The <Copy Colour Set> Button*: If the current interval number is lower than 21, the "Manual Colour Setting" dialogue box is opened, and the current settings are copied and can then be modified, otherwise the button is disabled.



The <Manual Colour Setting> Button

The <Manual Colour Setting> button opens the "Manual Colour Setting" dialogue box, where you can define the contour line values and the colour one wishes to assign to each interval. These settings can be saved as personalised colour profiles.



To change a colour, double click the coloured field and choose the new colour to be assigned to the interval from the Colours Dialogue Box.

Each time a colour set has been defined and confirmed, it is saved as a default manual colour profile. It will then be used each time the *Use manual colour settings* option is selected, if a different colour profile has not been loaded or saved; in this case, the default colour profile is overwritten, and the selected (saved or loaded) colour profile will be permanently associated to the current project, regardless of the default manual colour configuration.

- <New Profile> button: if the current interval number is lower than 21, the current automatic settings are copied and can then be modified, otherwise all the fields are cleared.
- <Read Profile> button: the program reads a previously saved colour profile from the hard disk; this colour profile will be permanently associated to the current project.
- <Save Profile> button: this allows one to save the current colour profile on the hard disk; this colour profile will be permanently associated to the current project.

Print Grid Command

With this command, the Print Preview window is opened and the plot is shown; it is then possible to directly print the graphical output, or export it to different formats.

The Graphical Output Preview Window

The Toolbar is located in the upper side of the preview window.

The Toolbar allows fast access to the commands with the mouse. Click the corresponding icon, to select one of the commands that belong to the Toolbar. The Toolbar displays the following commands:



- **Zoom +** : clicking the first icon activates the ZOOM function (it can only be used with the mouse) that permits the enlargement of part of the drawing. The function remains active until the Zoom - icon is selected. To enlarge a part of the drawing:
 - click the icon, and then select the area to be enlarged by clicking the upper-left corner and dragging the mouse pointer to the lower-right corner. At this point, release the mouse button. Note: due to the low resolution of the screen, the texts could appear in a slightly different scale at different zoom levels. This does not affect the quality of the final printouts.
- **Zoom -** : click this icon to return to the original scale of the preview.
- **Arrows**: click the arrow icons, in "Zoom +" mode, to pan the on-screen preview.

exaggeration factor.

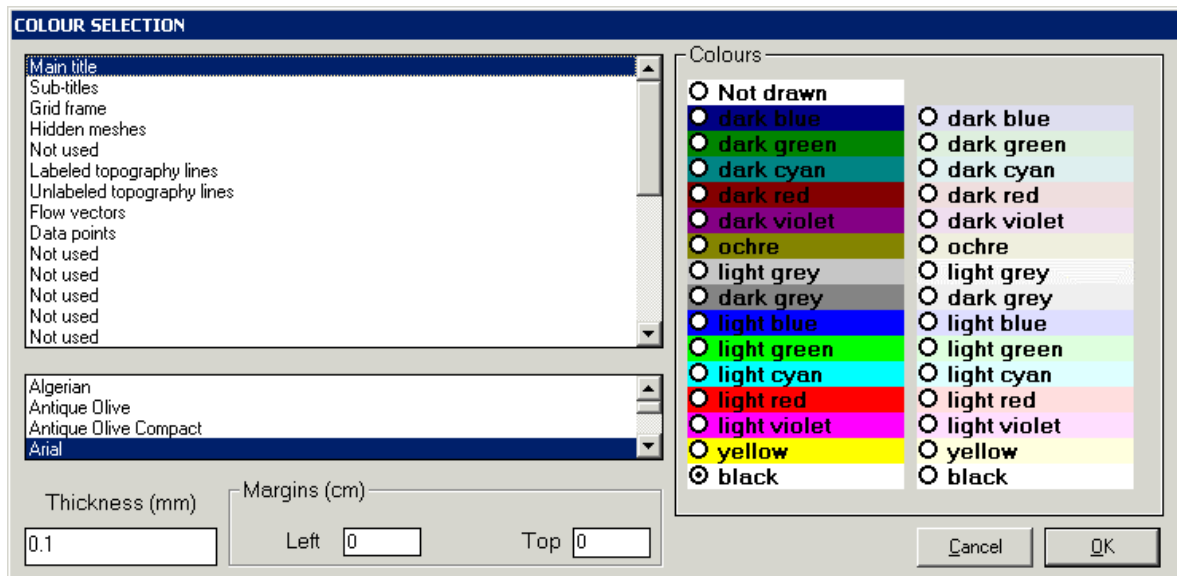
- Abscissa, Ordinate: the X and Y planimetric coordinates of the nodes of the line, from the arbitrary origin used to create the grid, along which the calculated points are aligned.
- Ok Button: the print preview window is shown, and the vertical section is plotted.
- Cancel Button: this is used to abort the current operation.

Export to SLK format Command

With this command, a window is shown to allow one to select which quantity has to be exported as an SLK file (a format that is compatible with Microsoft Excel and other spreadsheets).

Configure Command

With this command, one can define the graphical properties of the different objects in the graphical output.



One can associate a colour, a line thickness and a character font to each object. Notice that not all the objects have both a line thickness and a character font: for example, a *title* only requires the colour and the character font to be defined.

- Click an object to select it (use the scroll bar to see the entire list)
- Click one of the option buttons, in the “colours” frame, to select the colour that has to be associated to the selected object. The *Not drawn* option hides the object of the graphical output.
- If the object contains texts, one can choose a character font from the list (use the scroll bar to see the entire list)
- If the object contains lines, one can assign a thickness to the lines (in mm). If the value is zero, the program uses the thinnest line on the output device.
- It is possible, as an option, to assign a left and a top margin (in cm) to the whole graphical output.

NOTE: although the available set is limited to fifteen colours, they can be manually customised; double-click the coloured bar to open the Colour Dialogue Box and choose a different RGB value associated to the selected colour.

Exit Menu

This command allows one to exit the program. There is no specific command to save the entered data as these are automatically saved each time they are modified.

Chapter 8 - Slope Map

Slope Map (ISOMAP module)

With this command, the program creates a grid that contains the slope (in degrees) of a topographic surface. It is possible to choose whether the program has to calculate the maximum or the average slope of the ground around each node of the grid. As i , j are the grid node indexes and l is the mesh size, we obtain:

maximum slope =

$$\sqrt{\max(z(i+1, j) - z(i, j), z(i-1, j) - z(i, j))^2 / l^2 + \max(z(i, j+1) - z(i, j), z(i, j-1) - z(i, j))^2 / l^2}$$

average slope =

$$\sqrt{(z(i+1, j) - z(i-1, j))^2 / 4l^2 + (z(i, j+1) - z(i, j-1))^2 / 4l^2}$$

Procedure

- Select the "New Project" command from the "Project" menu
- Choose a new name for the project
- Select "Maximum Slope Map" or "Average Slope Map" in the "Operation" window
- Select the project that contains the original topographic surface
- Select "Calculation" on the menu bar

Edit menu - Slope Map

With this command, the program creates a grid that contains the slope (in degrees) of a topographic surface. When creating the new project, it is possible to choose whether the program has to calculate the maximum or the average slope of the ground around each node of the grid.



Hidden Mesh Selection Command

This command allows one to hide some of the grid meshes.

Calculate Menu - Slope Map

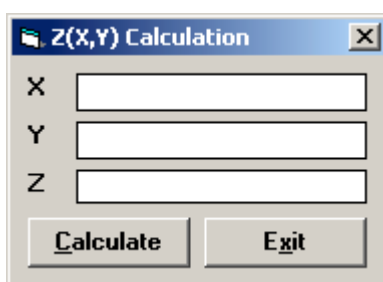
This menu allows one to calculate a grid that contains the slope of a topographic surface (in degrees).

Execute Calculations Command

This command allows one to calculate the slope of a topographic surface (in degrees).
Use the print menu to view the results of the elaboration.

Z(X,Y) Calculation Command

With this command, one can evaluate the "z" value of any point of the surface; enter the coordinates of the point in the window and click the <Calculate> button.



The program calculates the required value using a bi-cubic splines interpolation algorithm, and displays the result in the "Z" field.

Calculation of the volume Command

With this command, the programme calculates the volume (surface integral) of the whole grid, and shows it in a message box.

Print Menu - Slope Map

The Print menu is used to configure the graphical output and to plot the surface in Map, Wireframe View or Solid View form.

For more information, see the Print Menu - ISOMAP - Grid Generation Operation.

Chapter 9 - Exposure Map

Exposure Map (ISOMAP module)

With this command, the program creates a grid that contains the angle from the North of the dip direction of the ground. The Y-axis must be oriented to the North. The South direction will result to be equal to 180°; both the East and West directions will be equal to 90°.

Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Exposure Map” in the “Operation” window
- Select the project that contains the original topographic surface
- Select “Calculation” on the menu bar

Note: as an alternative, it is possible to obtain interesting results working on the original topographic surface using a horizontal Light Vector.

Edit menu - Exposure Map

With this command, the program creates a grid that contains the angle from the North of the dip direction of the ground. The Y-axis must be oriented to the North. The South direction will result to be equal to 180°; both East and West directions will be equal to 90°.



Hidden Mesh Selection Command

This command allows one to hide some of the grid meshes.

For more information, see the Hidden Mesh Selection in the Grid Generation Operation.

Calculate Menu - Exposure Map

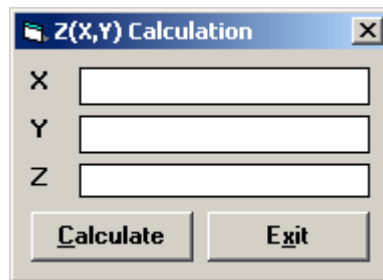
This menu allows one to calculate the angle from the North of the ground dip direction.

Execute Calculations Command

This command allows one to calculate the angle from the North of the ground dip direction. Use the print menu to view the results of the elaboration.

Z(X,Y) Calculation Command

With this command, one can evaluate the "z" value of any point of the surface; enter the coordinates of the point in the window and click the <Calculate> button.



The program calculates the required value using a bi-cubic splines interpolation algorithm, and displays the result in the "Z" field.

Calculation of the volume Command

With this command, the programme calculates the volume (surface integral) of the whole grid, and shows it in a message box.

Print Menu - Exposure Map

The Print menu is used to configure the graphical output and plot the surface in Map, Wireframe View, or Solid View form.

For more information, see the Print Menu - Grid Generation Operation.

Chapter 10 - Grid Difference

Grid Difference (ISOMAP module)

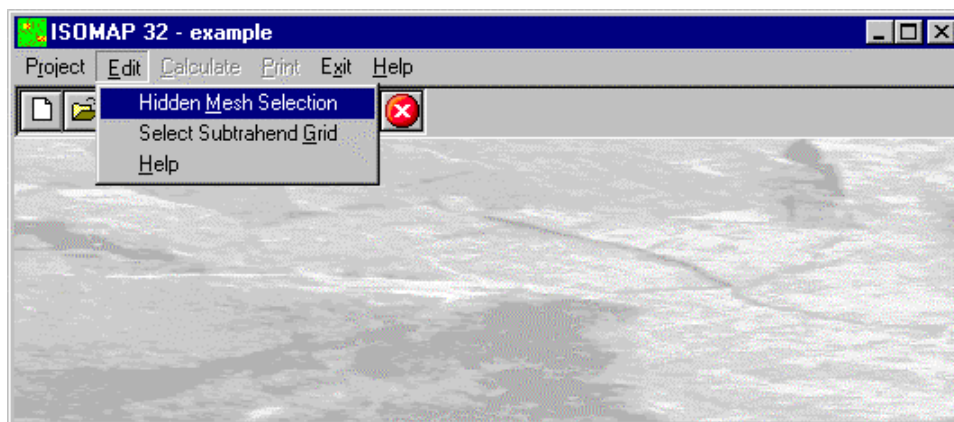
With this command, the program creates a new grid that is calculated as the node-by-node subtraction of two given grids. A typical application could be the evaluation of the removed ground, which is obtained by subtracting two grids that represent the topographic surface before and after an excavation. It is mandatory that the two grids have the same number of rows and columns.

Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Grid Difference” in the “Operation” window
- Select the project to extract from the “minuend” grid
- Select the “Select Subtrahend Grid” command from the “Edit” menu
- Choose an existing project (of the same grid size) to extract from the “subtrahend” grid
- Select “Calculation” on the menu bar

Edit menu - Grid Difference

With this command, the program creates a new grid that is calculated as the node-by-node subtraction of two given grids. A typical application could be the evaluation of the removed ground, which is obtained by subtracting two grids that represent the topographic surface before and after an excavation. It is mandatory that the two grids have the same number of rows and columns.



Hidden Mesh Selection Command

This command allows one to hide some of the grid meshes. For more information, see the Hidden Mesh Selection in the Grid Generation Operation.

Select Subtrahend Grid Command

This command allows one to choose an existing project (of the same grid size) to extract from the “subtrahend” grid

The “Select Subtrahend Grid” Dialogue Box

To select a file, type the name of the project to extract from the “subtrahend” grid in the File Name field, or browse the hard disk and double-click the project file name.

Calculate Menu - Grid Difference

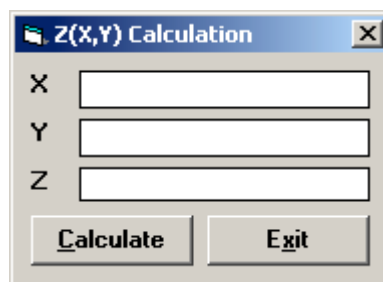
Once selected the Subtrahend Grid, this menu allows one to calculate the grid difference as the node-by-node subtraction of the two given grids.

Execute Calculations Command

Once selected the Subtrahend Grid, this command calculated the grid difference as the node-by-node subtraction of the two given grids. Use the print menu to view the results of the elaboration.

Z(X,Y) Calculation Command

With this command, one can evaluate the “z” value of any point of the surface; enter the coordinates of the point in the window and click the <Calculate> button.



The program calculates the required value using a bi-cubic splines interpolation algorithm, and displays the result in the "Z" field.

Calculation of the volume Command

With this command, the programme calculates the volume (surface integral) of the whole grid, and shows it in a message box.

Print Menu - Grid Difference

The Print menu is used to configure the graphical output and plot the surface in Map, Wireframe View, or Solid View form.

For more information, see the Print Menu - Grid Generation Operation.

Chapter 11 - Linear Transformation

Linear Transformation (ISOMAP module)

The linear transformation operator makes it possible to perform a linear transformation of the surface, performing the following three sequential operations for each node:

1. sums a first translation factor (a);
2. multiplies by a scale factor (b);
3. sums a second translation factor (c).

The general formula $z'=(z+a)*b+c$ makes it possible to perform any linear transformation such as the Celsius-Fahrenheit conversion, with $a=0$, $b=9/5$ and $c=32$.

Procedure

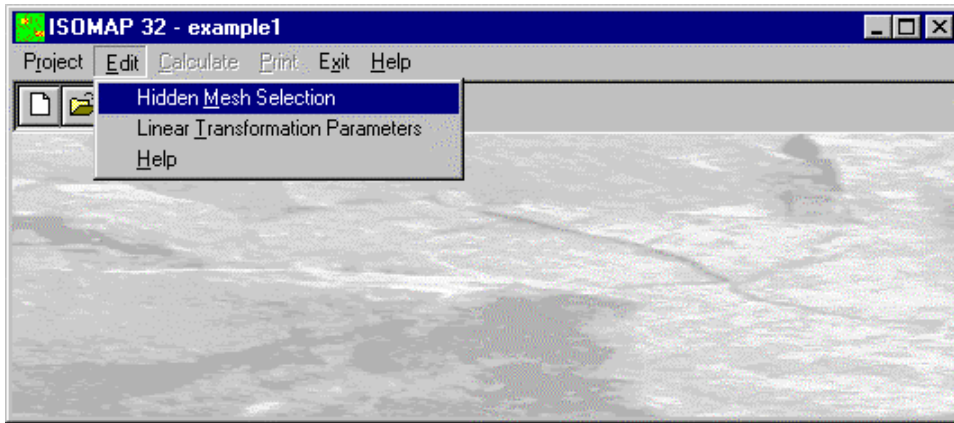
- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Linear Transformation” in the “Operation” window
- Select the project to be transformed
- Select the “Linear Transformation Parameters” command from the “Edit” menu
- Select “Calculation” on the menu bar

Edit Menu - Linear Transformations

The linear transformation operator makes it possible to perform a linear transformation of the surface, performing the following three sequential operations for each node:

1. sums a first translation factor (a);
2. multiplies by a scale factor (b);
3. sums a second translation factor (c).

The general formula $z'=(z+a)*b+c$ makes it possible to perform any linear transformation such as the Celsius-Fahrenheit conversion, with $a=0$, $b=9/5$ and $c=32$.



Hidden Mesh Selection Command

This command allows one to hide some of the grid meshes.
For more information, see the Hidden Mesh Selection in the Grid Generation Operation.

Linear Transformation Parameters Command

This command enables one to define the parameters for the linear transformation.

The “Linear Transformation Parameters” Dialogue box

In the “Linear Transformation Parameters” dialogue window the following data are required:

- *First translation factor*: it is the constant 'a' in the linear transformation $z'=(z+a)*b+c$ formula.
- *Scale factor*: it is the scale factor 'b' in the linear transformation formula.
- *Second translation factor*: it is the constant 'c' in the linear transformation formula.

Calculate Menu - Linear Transformation

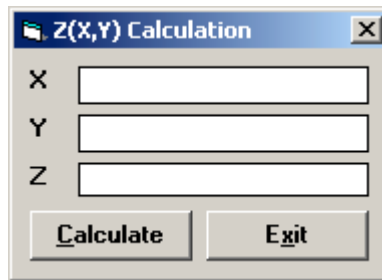
With this menu, a linear transformation is performed on each node of the source grid.

Execute Calculations Command

With this command, the linear transformation is performed on each node of the source grid, using the given parameters. Use the print menu to view the results of the elaboration.

Z(X,Y) Calculation Command

With this command, one can evaluate the “z” value of any point of the surface; enter the coordinates of the point in the window and click the <Calculate> button.



The program calculates the required value using a bi-cubic splines interpolation algorithm, and displays the result in the "Z" field.

Calculation of the volume Command

With this command, the programme calculates the volume (surface integral) of the whole grid, and shows it in a message box.

Print Menu - Linear Transformation

The Print menu is used to configure the graphical output and plot the surface in Map, Wireframe View, or Solid View form.

For more information, see the Print Menu - Grid Generation Operation.

Chapter 12 - Filtering

Filtering (ISOMAP module)

This operator performs a numerical filtering in the space domain, that is to say, the convolution of a matrix operator of order $2n+1$ with the grid itself.

The filters should always be symmetrical to the axis that passes through its centre, and to the two diagonals. By using a unit matrix one will obtain a grid that is the moving average of the original grid. By using this command in conjunction with the Grid Difference operation, one can, for example, separate the local gravity anomalies from the regional ones.

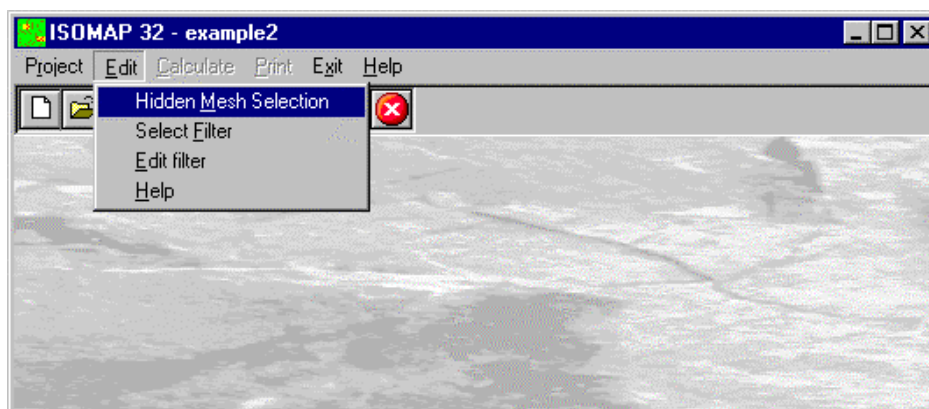
Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Filtering” in the “Operation” window
- Select the project to be filtered
- Select the “Select Filter” command from the “Edit” menu
- Choose a filter (you can create a new one with a text editor)
- Select “Calculation” on the menu bar

Edit Menu - Filtering

This operator performs a numerical filtering in the space domain, that is to say, the convolution of a matrix operator of order $2n+1$ with the grid itself.

The filters should always be symmetrical to the axis that passes through its centre, and to the two diagonals. By using a unit matrix one will obtain a grid that is the moving average of the original grid. By using this command in conjunction with the Grid Difference operation, one can, for example, separate the local gravity anomalies from the regional ones.



Hidden Mesh Selection Command

This command allows one to hide some of the grid meshes.
For more information, see the Hidden Mesh Selection in the Grid Generation Operation.

Select Filter Command

With this command, the program shows the “Select Filter” dialogue box.

The “Select Filter” Dialogue box

To select a file, type the name of the file containing the digital filter to be used for the current operation in the File Name field, or browse the hard disk and double-click the filter file name.

To create a new digital filter, type a new name and confirm, and then select the command Edit Filter from the Edit menu.

Edit Filter Command

With this command Notepad is started, and one can create or edit a custom filter.
The structure of the filters is quite simple: the order of the matrix operator is written in the first line. This should always be an odd number. The coefficients are written in the following lines, separated by spaces.

This is an example of a filter that performs the average of the values along a circle centred on a grid node:

```
11
.00000 .00000 .00000 .03571 .03571 .03571 .03571 .03571 .00000 .00000 .00000
.00000 .00000 .03571 .00000 .00000 .00000 .00000 .00000 .03571 .00000 .00000
.00000 .03571 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .03571 .00000
.03571 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .03571
.03571 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .03571
.03571 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .03571
.03571 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .03571
.00000 .03571 .00000 .00000 .00000 .00000 .00000 .00000 .00000 .03571 .00000
.00000 .00000 .03571 .00000 .00000 .00000 .00000 .00000 .03571 .00000 .00000
.00000 .00000 .00000 .03571 .03571 .03571 .03571 .03571 .00000 .00000 .00000
```

The filters can be created with any external word-processor, should have the “.FIL” extension and should be saved as plain text.

Calculate Menu - Filtering

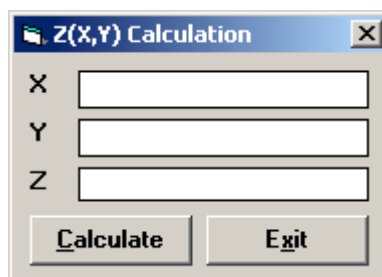
With this menu, the grid filtering is performed.

Execute Calculations Command

With this command, the grid filtering is performed using the chosen filter.

Z(X,Y) Calculation Command

With this command, one can evaluate the “z” value of any point of the surface; enter the coordinates of the point in the window and click the <Calculate> button.



The program calculates the required value using a bi-cubic splines interpolation algorithm, and displays the

result in the "Z" field.

Calculation of the volume Command

With this command, the programme calculates the volume (surface integral) of the whole grid, and shows it in a message box.

Print Menu - Filtering

The Print menu is used to configure the graphical output and plot the surface in Map, Wireframe View, or Solid View form.

For more information, see the Print Menu - Grid Generation Operation.

Chapter 13 - Grid Duplication

Grid Duplication (ISOMAP module)

This command enables one to duplicate a project, for example to allow parametric analyses. It can also be used to create a new grid from an existing one, changing the extension of the grid and the size of the mesh.

Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Grid Duplication” in the “Operation” window
- Select the project to be duplicated
- Assign the new values of Xmin, Xmax, Ymin, Ymax and of the Mesh Size
- Select “Calculation” on the menu bar

Note: the X,Y values must be inside the ranges of the original grid.

Chapter 14 - DTM Import

DTM Import (ISOMAP module)

With this command, it is possible to import regular grid data from other programs and convert these into the ISOMAP format. A default ISOMAP format can be used, or a custom made filter can be developed on request. The data files must previously have been created, with a text editor such as Notepad, and have the “.ZRE” extension.

The structure of the ZRE files is quite simple: on the first line write the number of nodes on the X-axis, on the second line write the number of nodes on the Y-axis, on the third line write the mesh size (in metres) between two nodes, in the two next lines write the abscissa and ordinate of the first grid node in the bottom left corner. The file can now be completed with the Z-values, column by column (left to right), each column in a bottom-up order. Each column of Z-values must be preceded by a line that contains the text “Column #”, where # is the column number. Once the file is saved, the import procedure can be started.

Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Import ZRE file” in the “Operation” window
- Choose the ZRE file you want to import the grid from

Once confirmed the file name, the programme executes the conversion, then it directly activates the Print menu, because no further calculation must be performed.

Chapter 15 - VID Import

Import VID File (ISOMAP module)

This command enables one to import a file in VID format.

Procedure

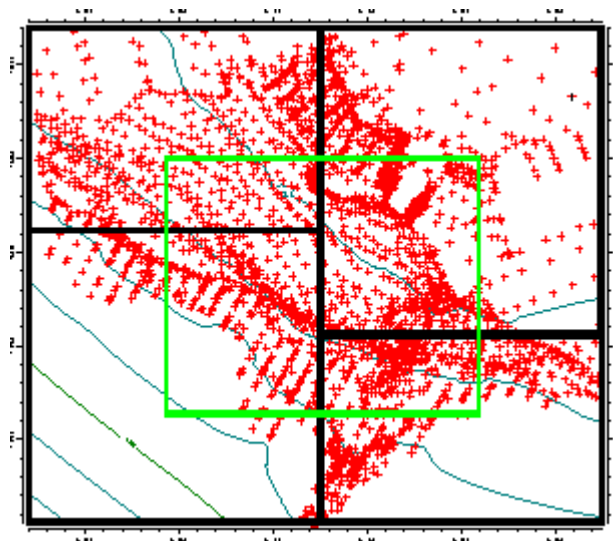
- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Import VID file” in the “Operation” window
- Choose the VID file you want to import the grid from

Once confirmed the file name, the programme executes the conversion, then it directly activates the Print menu, because no further calculation must be performed.

Chapter 16 - XYZ Import

Import XYZ file (ISOMAP module)

This command enables one to manage XYZ files. It is possible to join two or more files and, eventually, extract from them a partial area. This feature can be particularly useful when the area of interest (the green rectangle in the picture) crosses different, individual surveys (the black rectangles).



Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Import XYZ file” in the “Operation” window
- Type the coordinates of the lower-left and upper-right corners of the new grid
- Type XYZ file names you want to import the points from

Once confirmed the file names, the programme executes the conversion, then it activates the Calculation menu.

Chapter 17 - DXF Import

Import DXF file (ISOMAP module)

This command enables one to import a file in DXF format.

To correctly import a DXF file, it must have been saved as **Autocad release 12** file, the contour lines must be POLYLINES (LINES are not imported), and the elevation value of each polyline must have been embedded in the file.

The screenshot shows the 'Files DXF Import' dialog box. It features a title bar with the text 'Files DXF Import' and a 'Find DXF' button in the top right corner. The main area is divided into two columns. The left column contains several input fields and buttons: 'Coastal lines layer' with an 'Import' button, 'Contour lines layer' with an 'Exit' button, 'Mesh size', 'Point Distance', and 'Z water'. Below these are pairs of input fields for 'X min' and 'Y min', 'X max' and 'Y max', and 'X water' and 'Y water'. The right column is a large, empty white rectangular area.

The programme extracts polylines only from the declared layers ("Contour lines layers" fields), so those layers must contain all and only the polylines representing contour lines (so roads, buildings and any other object in form of polyline must be removed from the declared layers before processing the file).

Once obtained a clean DXF file, the names of the layers (1 to 7 max) can be typed in the ("Contour lines layers"

fields.

Some of the other fields in the window are mandatory:

- Mesh size: width (and height), in meters, of the square meshes of the grid which will be created. Remember the maximum grid size is 500x500 meshes;
- X,Y coordinates of the lower-left (Xmin, Ymin) and of the upper-right (Xmax, Ymax) corners of the new grid. Those four values must be exact multiples of the mesh size.

The "Point Distance" field is optional, but strongly suggested for speed and quality reasons. This value allows the programme to pick few points out of the polylines, which are often very densely populated, obtaining a more isotropic point distribution which brings to a faster and more robust elaboration.

A good value for this parameter is the average planimetric distance between the contour lines.

FREE WATER TABLES

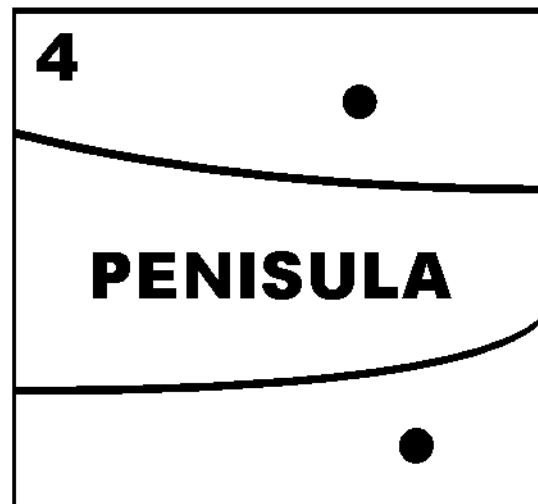
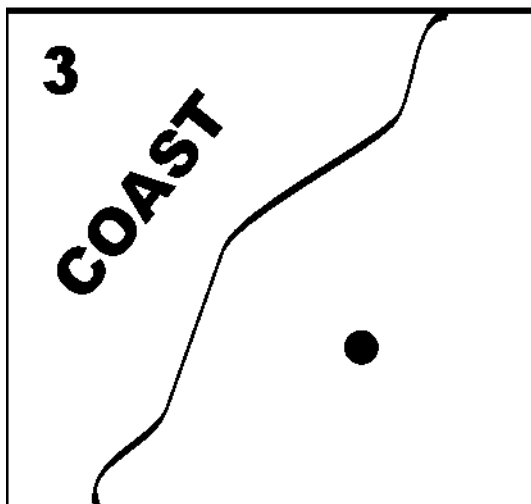
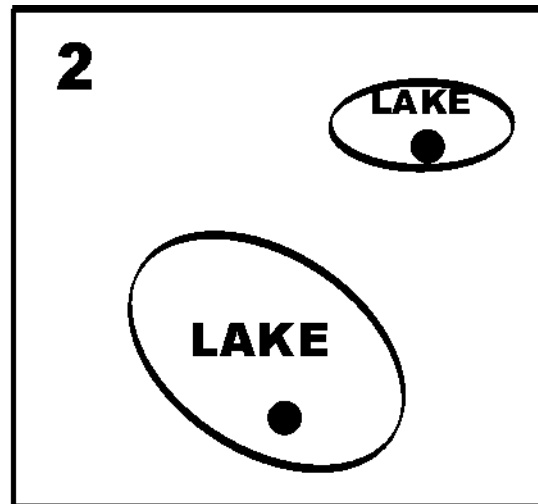
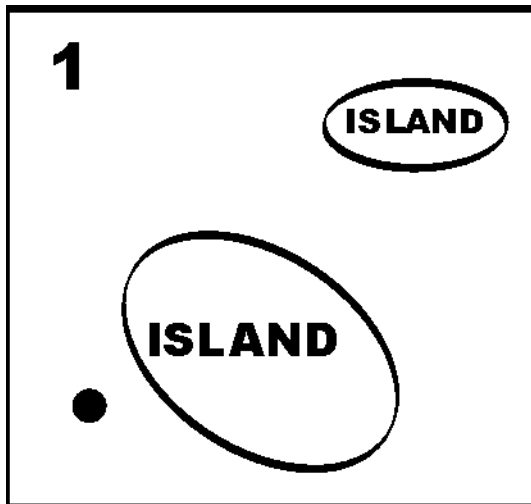
To get a more agreeable output in presence of lakes or sea, it is possible to use a simple trick to assign a constant value to the water surfaces.

Before importing the file, it is enough to add to the DXF file one or more coastal lines (see figure below), taking care that the closed lines are really closed (i.e. the first and the last points have the same coordinates) and the crossing lines surely exceed the rectangle defined by the lower-left and upper-right points previously typed.

Those lines must be placed on a dedicated layer.

Now we can compile the "Z water" field, typing the water table elevation; for graphical reasons, we suggest to choose a value slightly lower than the real value if this is an exact multiple of the contour line distance: for example, for the sea, use -0.1 instead of 0, to avoid generating random horizontal lines.

For each separated water table, a couple of coordinates "X water" and "Y water" must be typed, to tell the programme which side of the coastal lines the water is; for example, you could give the coordinates of the big black points in the figure below.



Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Import DXF file” in the “Operation” window
- Choose the DXF file you want to import the grid from, and compile the required fields

Once confirmed, the programme executes the conversion, then it activates the Calculation menu.

Chapter 18 - ASC Import

Import ASC file (ISOMAP module)

This command enables one to import a file in ASC format.

Procedure

- Select the “New Project” command from the “Project” menu
- Choose a new name for the project
- Select “Import ASC file” in the “Operation” window
- Choose the ASC file you want to import the grid from

Once confirmed the file name, the programme executes the conversion, then it directly activates the Print menu, because no further calculation must be performed.