CLU_STAR
Cluster and Rock Stability Analysis

Summary

Chapter 1 - Introduction to the program

Introduction to the program

CLU-STAR is a program for computerised structural geology data collection and analysis. The program recognises the discontinuity sets of a rock mass through hierarchical and non-hierarchical clustering procedures derived from the multivariate analysis. These clustering procedures operate directly on the collected data instead of on their statistical distribution; the results are consequently closer to the geomechanical reality.
The high interactivity of the program offers the operator a better control of the full sequence of operations, as it is possible to choose the clustering level (and hence the dimension of the clusters), as well as the algorithms for the assignment of the joints to the individual discontinuity sets.
Once the joint sets have been determined, the program calculates the safety factor of the rock wedges, considering water pressures, surcharges, rock bolting and seismicity. The program can also evaluate the anisotropic permeability tensor of the rock mass.
Different outputs are produced both in graphical and tabular form:
- the printout of the structural geology data collection sheets;
- the projection of the originally collected joint poles onto the stereonet;
- complete projection of the results of the cluster analysis: the joint sets, the great circles, the rock wedge safety factors, the slope face and the friction angle circle can be represented on the stereonet;
- a rose diagram for the evaluation of the dip direction distribution.

Calculation methods: hierarchical and non-hierarchical clustering

The program recognises the discontinuity sets of a rock mass through hierarchical and non-hierarchical clustering procedures that are derived from the multivariate analysis.

Hierarchical Clustering

Hierarchical clustering means that a tree is built through a process of cluster aggregation; starting with *n* clusters, each of which contains only one pole, the two closest clusters are aggregated into one at each step and, after *n* steps, only one cluster is obtained that contains *n* poles.

This procedure offers three methods to calculate the distance between two clusters:

- the Group Mean method in which the distance is equal to the mean of all the distances between all the pairs of the poles of the two clusters.

- the Nearest Neighbour method in which the distance is equal to the distance between the two nearest poles of the clusters. This method gives rise to "dispersed" clusters, and shoud not be used for stability analyses.

- the Furthest Neighbour method in which the distance is equal to the distance between the two furthest poles of the clusters. This method gives rise to "compact" clusters, and is perhaps the most suitable for stability analyses.

The operator can then analyse the tree and choose the optimal clustering level.

Outsiders

In the geological history of a rock mass, a sequence of different tectonic events can lead to the formation of different discontinuities, such as joints and faults. In a rock wedge stability analysis, performed on a slope of a

given height, one assumes that two discontinuities exist, whose intersection passes through the toe of the slope. This can only occur if a sufficiently large number of discontinuities belongs to the two joint sets.

The program allows one to discard, in the wedge stability analyses, the clusters that only contain a small number of discontinuities. These un-clustered joints are called *outsiders*, and should be individually examined to verify whether they really can be considered as a "background noise", or if they are sufficiently important to need a separate analysis.

Optimal clustering level

The cluster size and number are inversely proportional. The criterion to choose the optimal clustering level changes according to the problem that has to be solved. In a large-scale tectonic analysis it could be important to identify a small number of clusters, regardless of how dispersed the joints around the cluster centre are. On the other hand it is fundamental, in the stability analyses, to keep the joints close to the cluster centres as the cluster dispersion has a great influence on the results of the stability calculations.
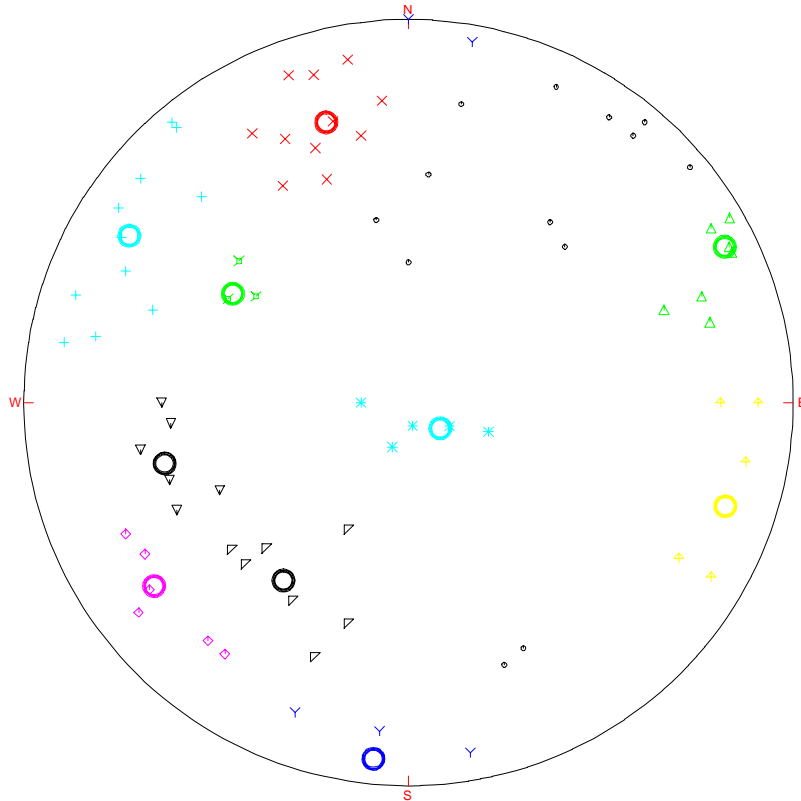
figure1
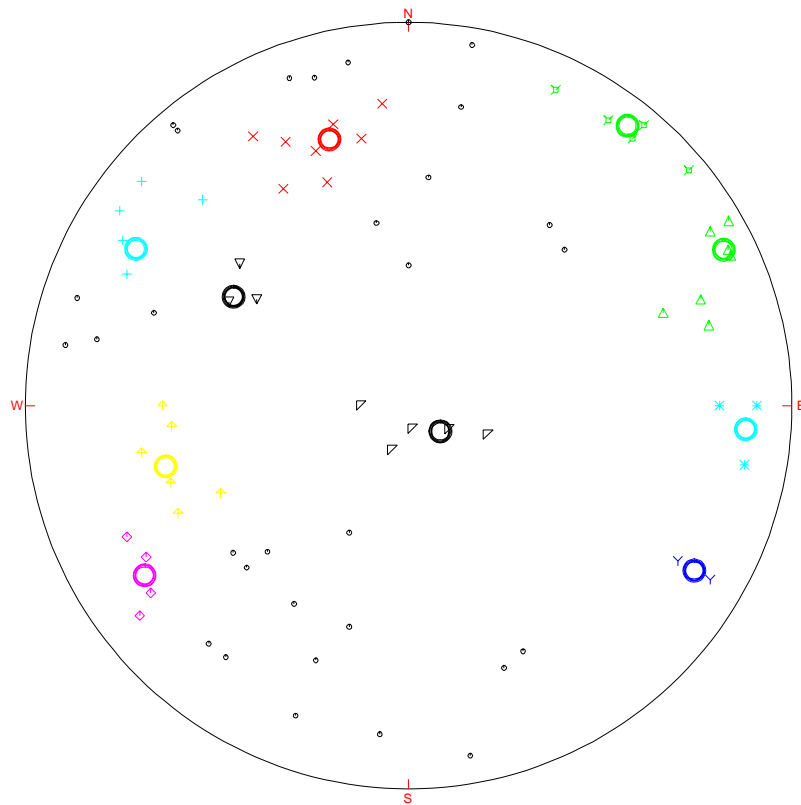
figure 2

The program allows one to select the largest clusters from those that have been obtained by choosing a certain clustering level. This allows one to identify the most important joint sets, which result regardless of the initially chosen clustering level.

For example the ten largest clusters are shown in the figures 1 and 2. These have been chosen from the two initial clustering levels of 15 and 23 joint sets. One can note that only some small and dispersed clusters are affected by the initial clustering conditions in these two analyses.
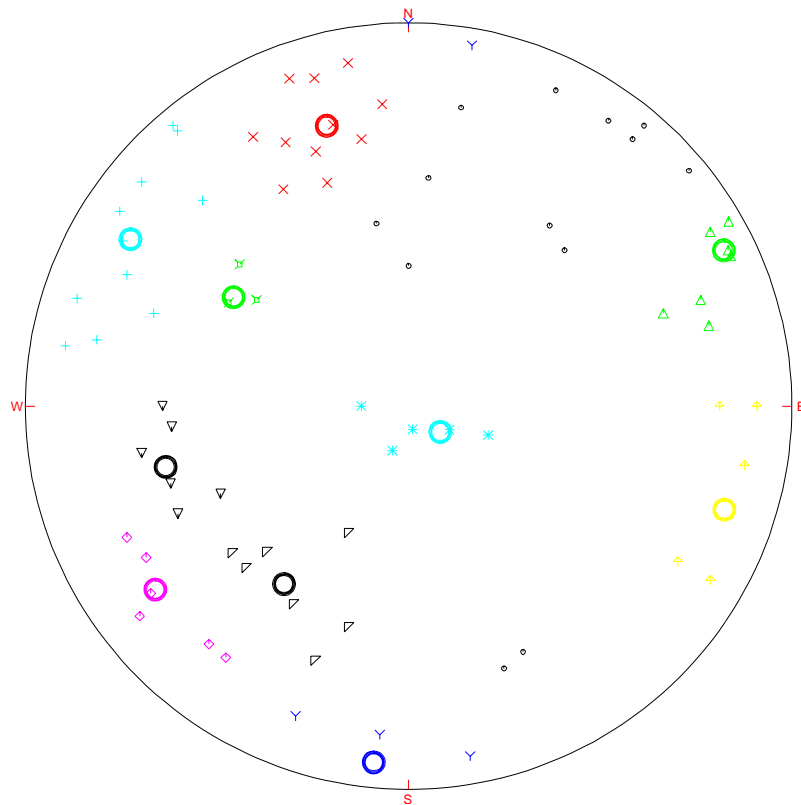
figure 3
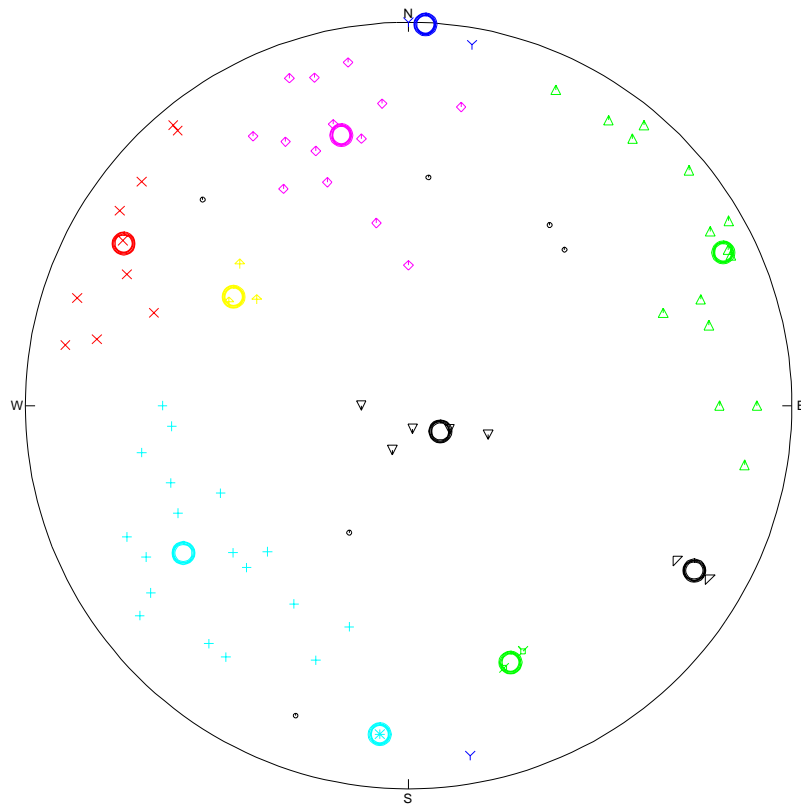
figure 4
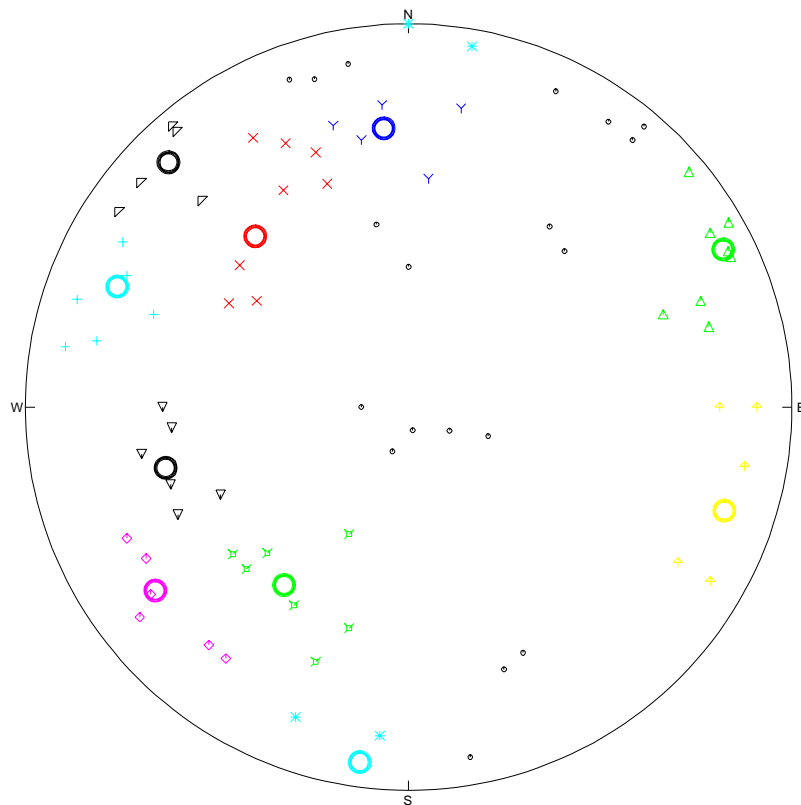
figure 5

Figures 3, 4 and 5 illustrate how the cluster distribution could change when using different methods to calculate the distance. Group Mean, Nearest Neighbour and Furthest Neighbour methods have been used, and the 10 largest clusters have been chosen from an initial group of 15 joint sets. Again, only the smallest and most dispersed clusters are affected by the change of the method used to calculate the distance.
Using and comparing different distance calculation methods and initial clustering levels could be considered as an efficient method to validate the obtained results.

Non-Hierarchical Clustering

An optional stage of non-hierarchical clustering can be performed to refine the results obtained with the hierarchical clustering. This method is based on the following steps:

1. the operator defines a limit angle: a pole can only belong to a cluster if the angular distance from the cluster centre is lower then the given angle.

2. the positions of the cluster centres are calculated as the weighted vectorial sum of the poles that belong to the clusters.

3. the poles are reassigned to the clusters taking the limit angle and the new cluster centre position into consideration.
Steps 2 and 3 are automatically repeated until a stable configuration is reached.
Non-hierarchical clustering allows one to optimise the positioning of the cluster centres.
The vectorial sum of the pole direction vectors disregards the density distribution of the poles. When instead a suitable value of the limit angle is used, the centre of the clusters moves towards the more dense area.
The effects of non-hierarchical clustering, with limit angles of 30-20-10 degrees, are shown in figures 6, 7 and 8. When the limit angle value is decreased, the number of outsiders increases, while the position of the most dense area of the clusters is identified with greater precision.
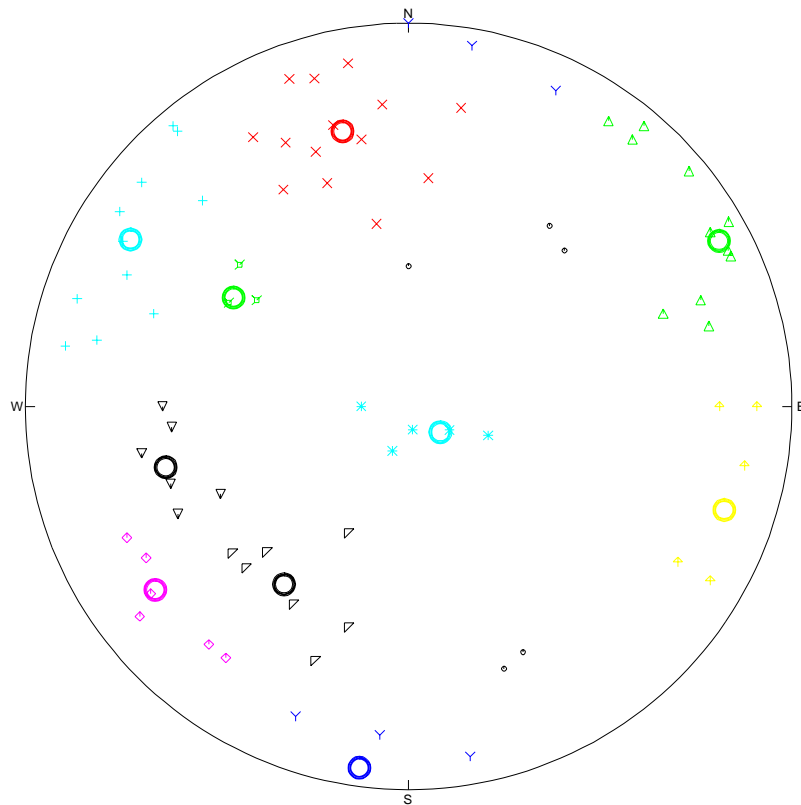
figure 6
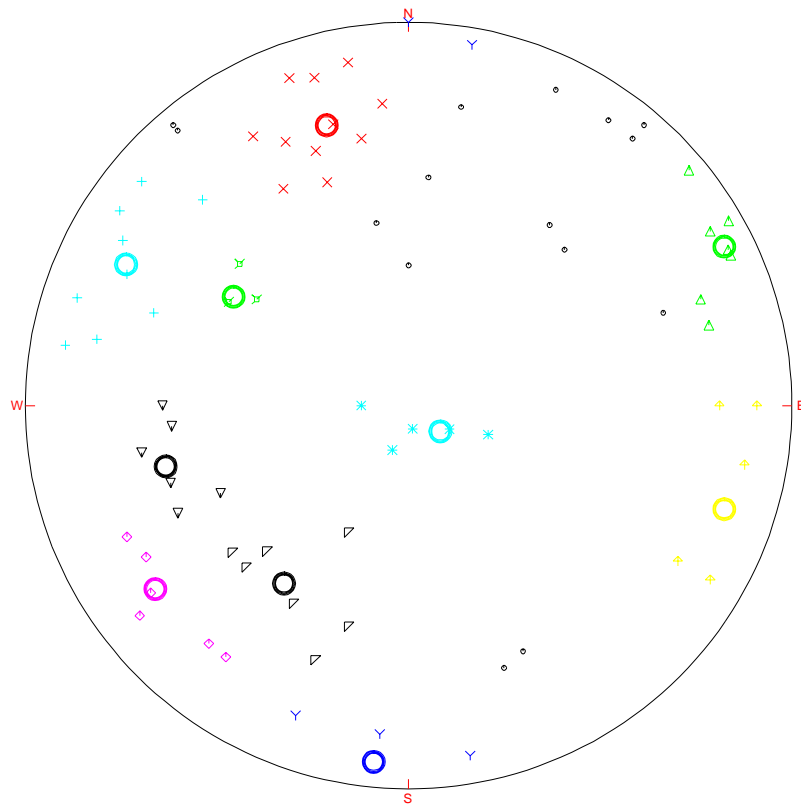
figure 7
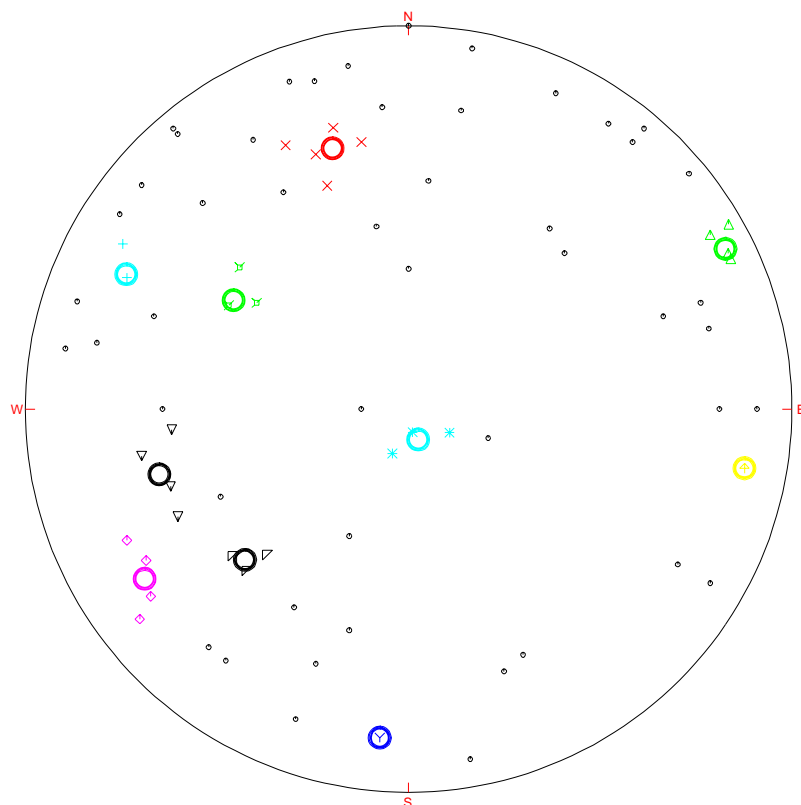
figure 8

Rock Wedge Stability Analysis

The rock wedge stability analysis is performed using the comprehensive solution published in E. Hoeck, J.W. Bray, "Rock Slope Engineering", 1981, The Institution of Mining and Metallurgy, London.
This solution is for the computation of the factor of safety for translational slip of a tetrahedral wedge formed in a rock slope by two intersecting discontinuities, the slope face and the upper ground surface. The influence of a tension crack is included in the solution. The solution does not take account of rotational slip or toppling.

The solution allows for different strength parameters and water pressures on the two planes of weakness and for water pressure in the tension crack. There is no restriction on the inclination of the crest of the slope. The influence of an external load and a cable tension are included in the analysis.

Reference can be made to "Rock Slope Engineering" for further detail on the mathematical development of this solution.

Anisotropic Permeability Tensor Calculation

Given a single joint set, the permeability parallel to the discontinuities is given by:

$$K = (g\, e^3) / (12\, v\, b)$$

where:

g = gravitational acceleration
e = opening of cracks or fissures
b = spacing between cracks
v = coefficient of kinematic viscosity

This permeability value is isotropic in all the directions parallel to the discontinuity plane, and is null in the direction of the vector that is normal to this plane. The permeability can thus be seen as a tensor, and the permeability of the whole rock mass is calculated, in terms of its main components, as a direct tensor sum of the permeability tensors of all the joint sets.

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 <u>must</u> be installed, even if the printer itself is not connected to the PC.**

Program installation

To install the CLUSTAR program, run **CLUSTAR32SETUP.EXE** from the CD-ROM or from the folder where you downloaded and saved the setup program.
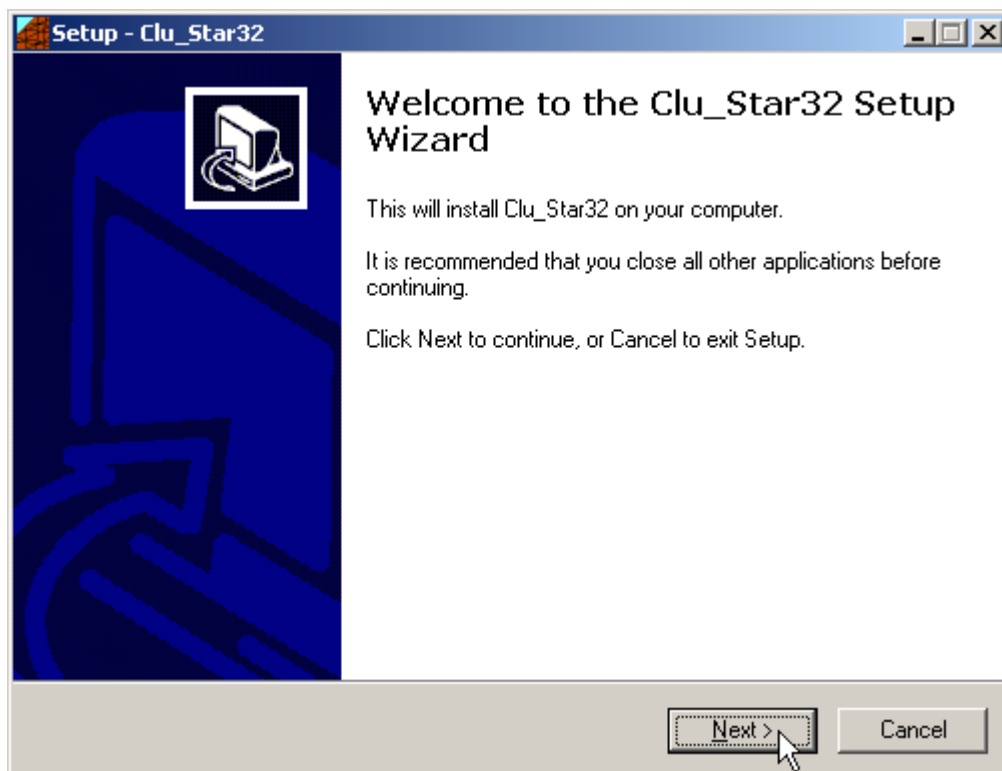
Press the **NEXT** button to start the installation.
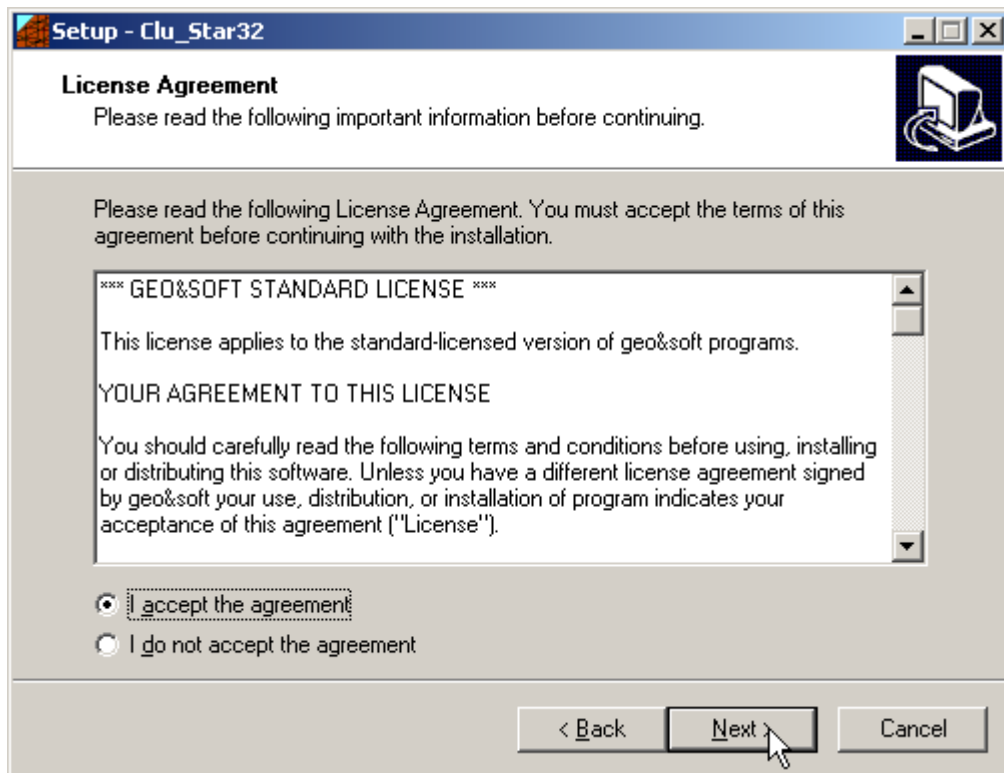
Carefully read the License Agreement and press the **NEXT** button.
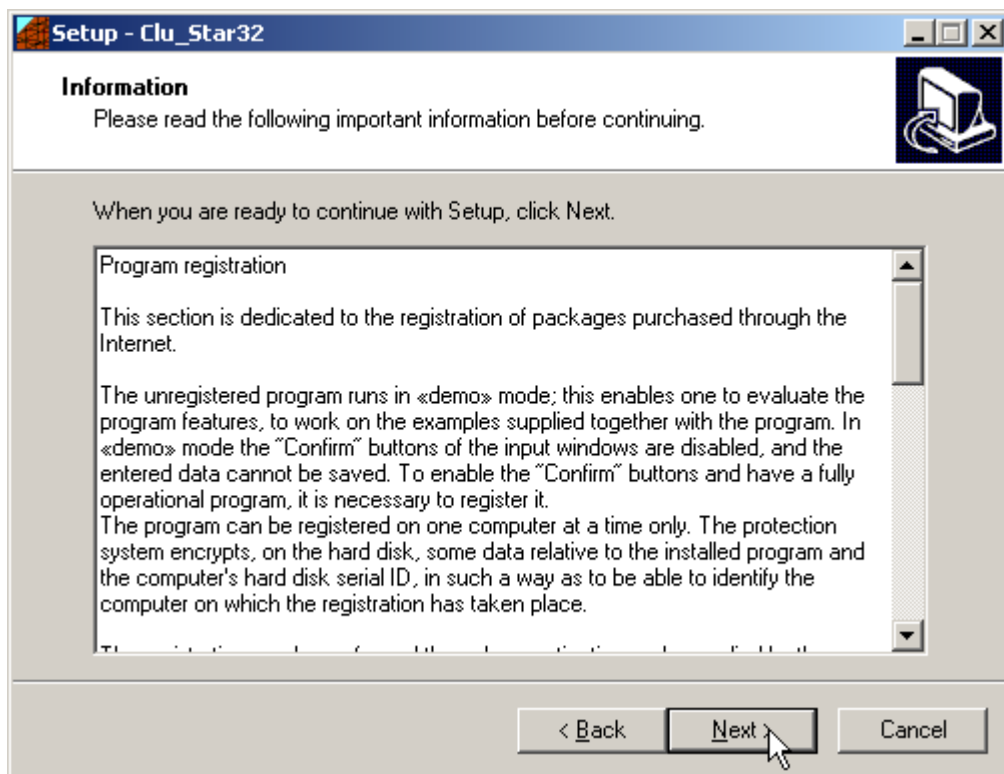
Carefully read the informations about the registration of the software and press the **NEXT** button.
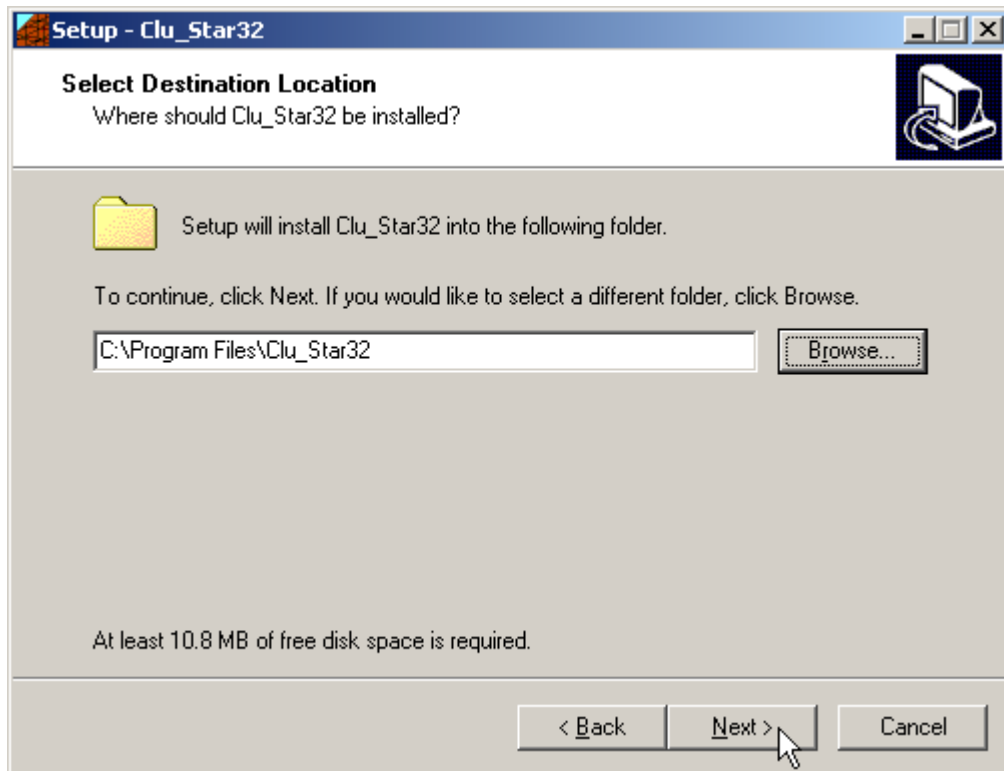
Select the folder where the **CLUSTAR32** directory would be created and press the **NEXT** button.
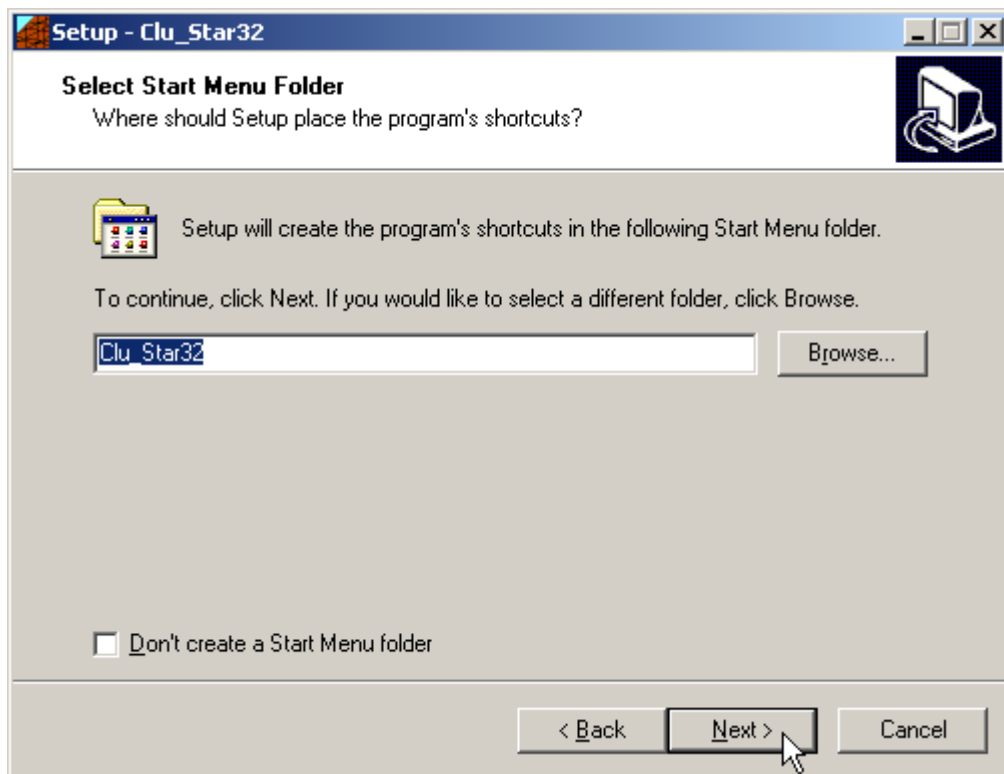
Press the **NEXT** button to create the **CLUSTAR32** group in the Start menu.
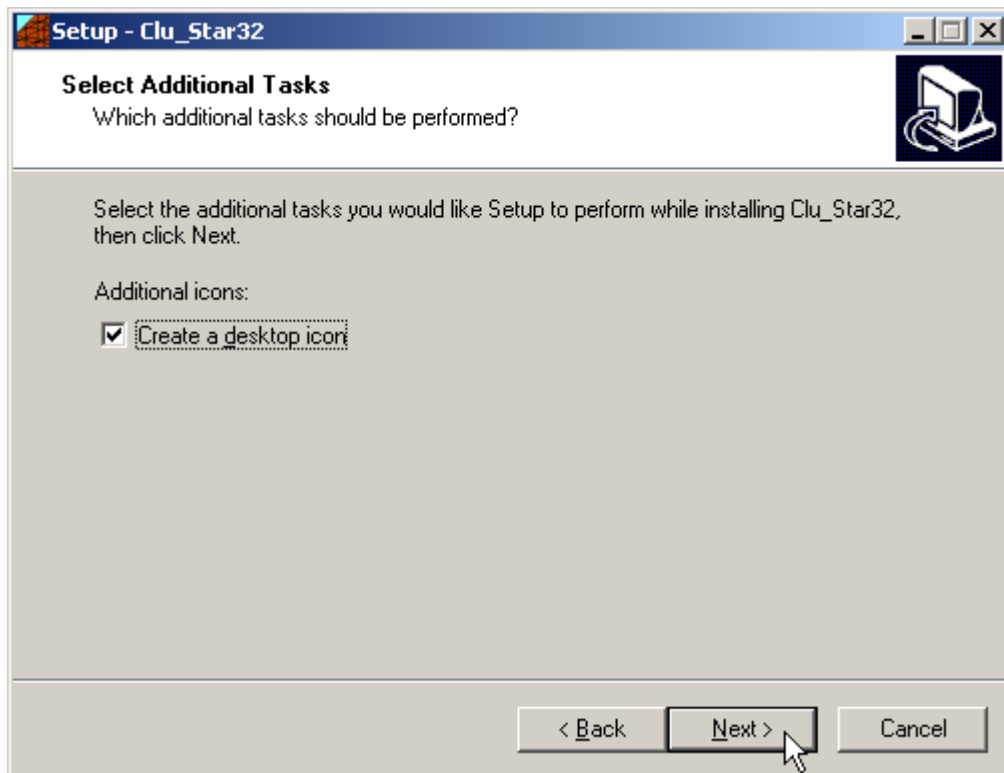
Choose if you want an icon to be created on your desktop, or not.
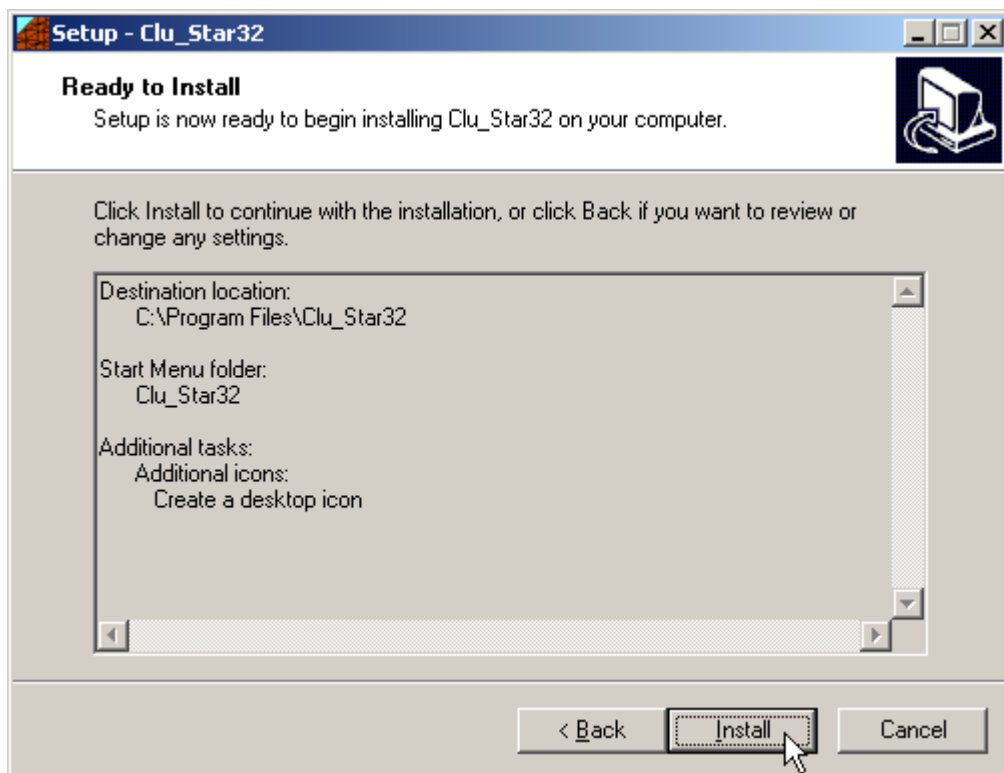
Press the **INSTALL** button to continue the installation procedure …
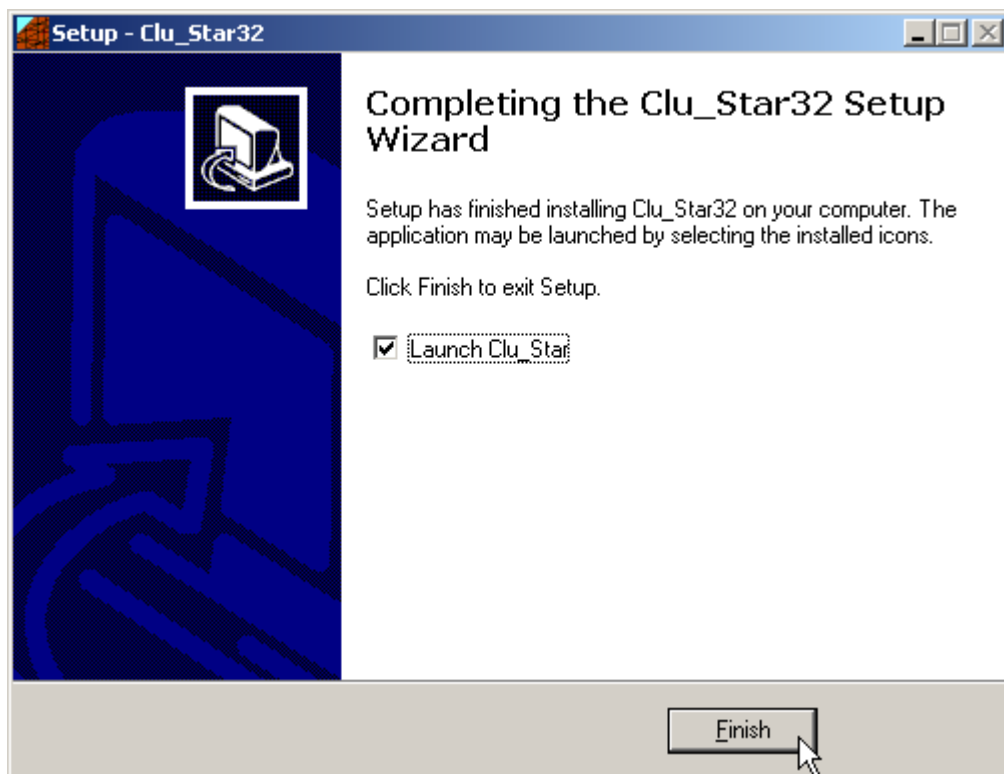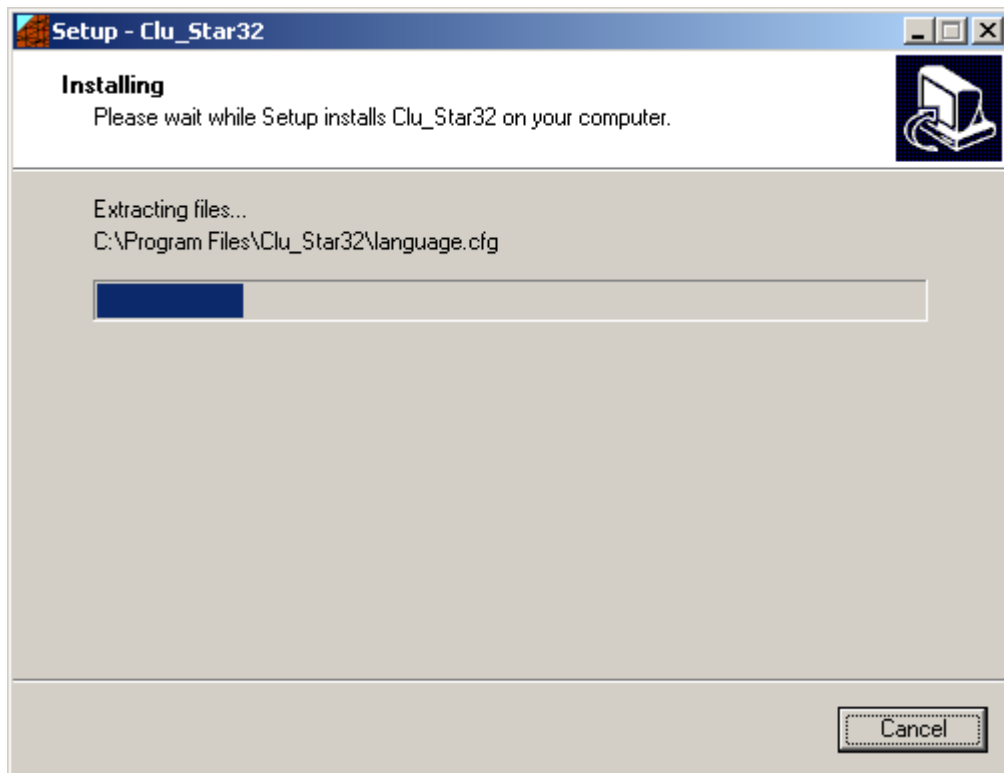
Once completed the installation, press the **END** button to exit the setup procedure and to start **CLUSTAR32**.

CLUSTAR32 can be run by clicking the **Start** button, then selecting the **CLUSTAR32** group in the **Programs** menu, and clicking **CLUSTAR32.**

Chapter 3 - The program protection

Program registration

This section is dedicated to the registration of packages purchased through the Internet.
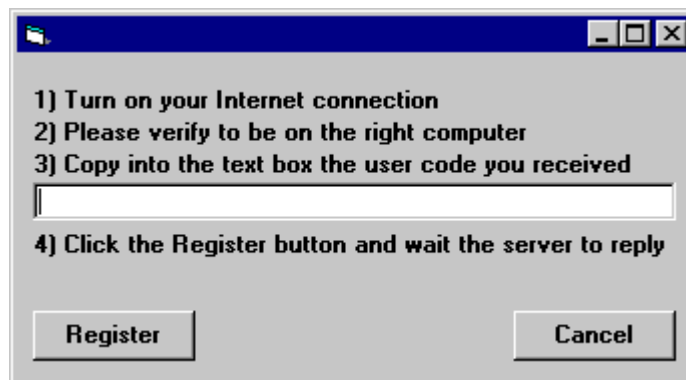
The unregistered program runs in «demo» mode; this enables one to evaluate the program features, working on the examples supplied together with the program. In «demo» mode the "Confirm" buttons of the input windows are disabled, and the entered data cannot be saved. To enable the "Confirm" buttons, and 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:
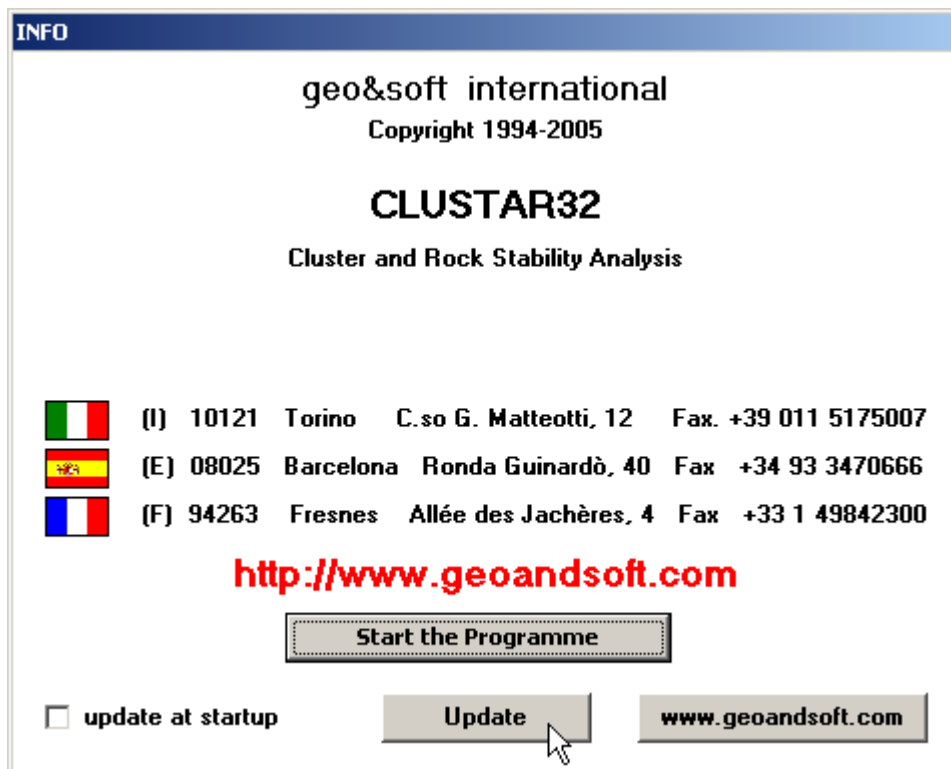
1. Start the program

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.
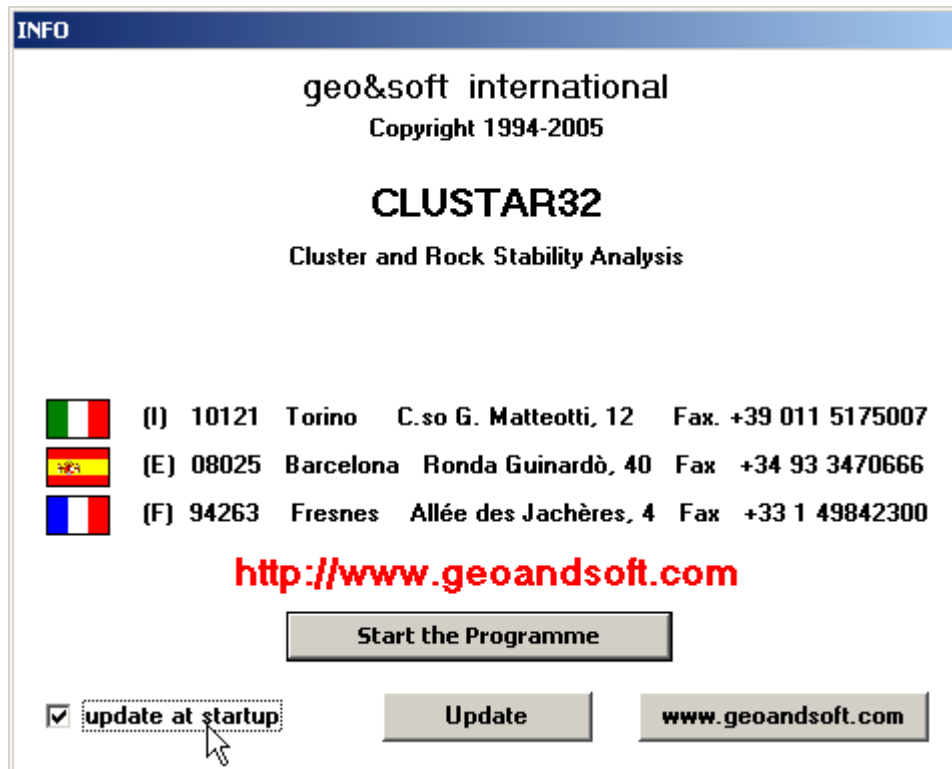
Autoupdating requirements and troubleshooting

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.
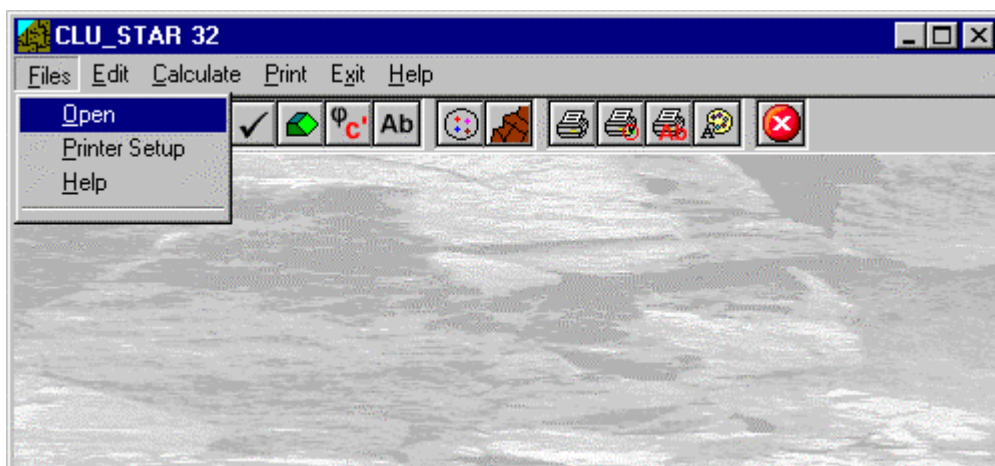
The programs automatically disable some menus or commands when the execution of such menus or commands is not yet feasible. Typical examples are: a calculation menu is disabled when not all the data needed for the calculation have been completely entered, or a print menu is disabled if not all the calculations 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.
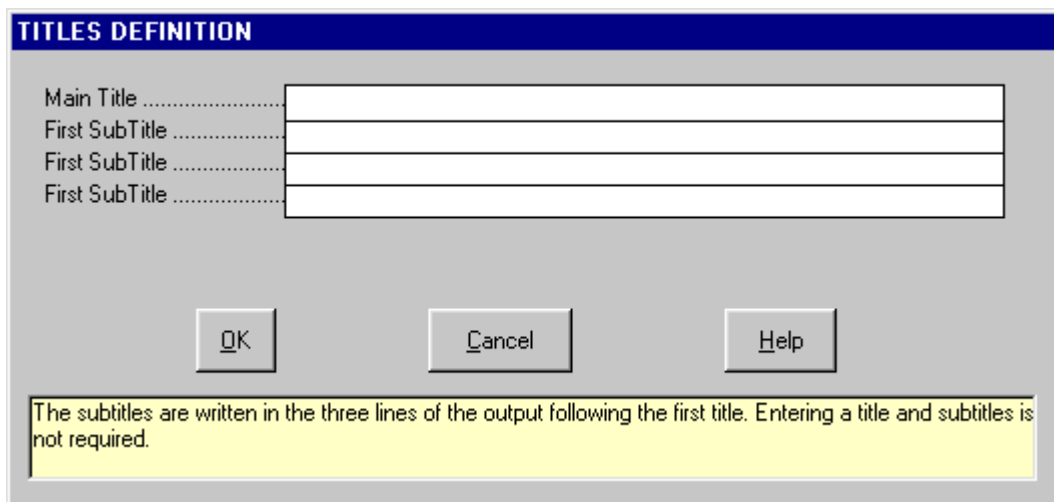
Some fundamental tools are used inside the dialogue windows: data fields such as text and list boxes, and buttons.

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.

Tere 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:
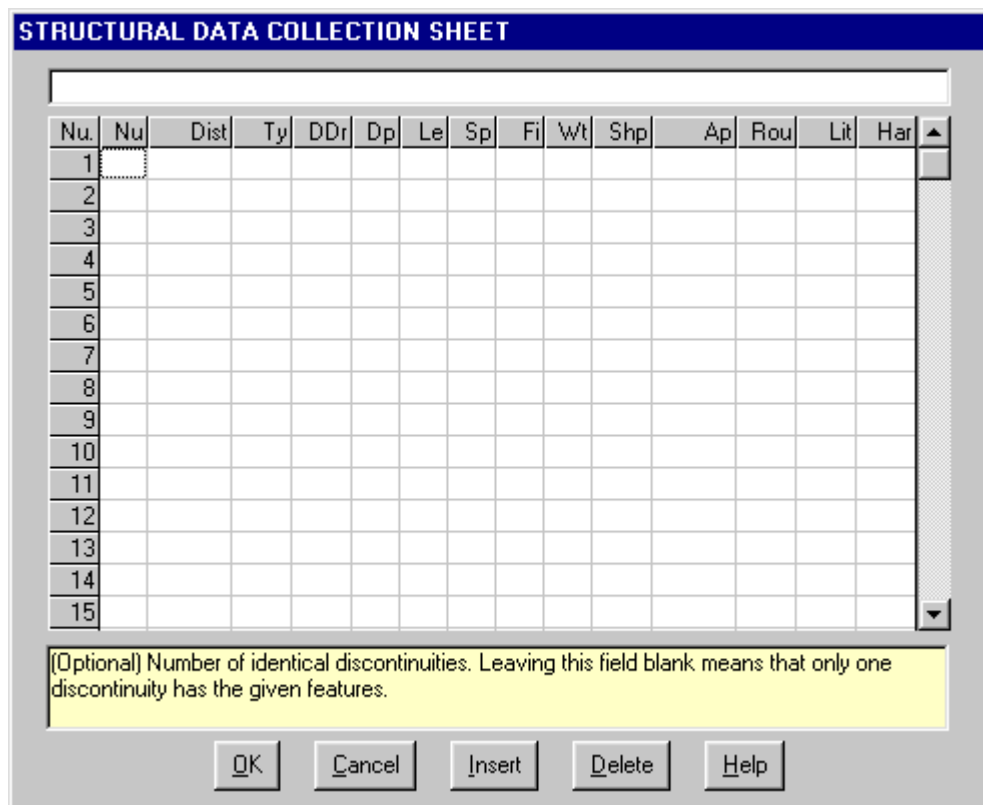
- TAB - moves the cursor to the first button in the window. When one presses this key again, the cursor is moved to the next button.

- 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.
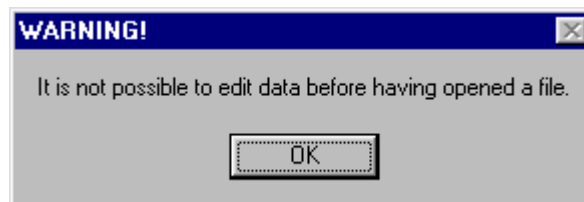
Help On Line

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 - Commands

Files Menu

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

Open 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 file:

☞   choose the Open command from the menu by first clicking the Files menu item and then on the Open 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 ".SES" is automatically added: for example, given the name "TEST01", the file name "TEST01.SES" will be internally used by the program.

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

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.

Edit Menu

This menu is used to enter and edit the input data used for the cluster and rock wedge stability analysis.

Structural Data Collection Command

This command enables one to enter and edit the geometrical and geomechanical data acquired with the structural geology data collection. In order to edit the structural data, it is necessary to open a new or existing sheet. The program allows one to collect data from different sites, to store them as separate files, and to perform the cluster analysis on the data collected in one or more sheets at the same time.

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

✍   choose the Open command from the menu by first clicking the Files menu item and then on the Open command. Type the name of the sheet 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 ".SCH" is automatically added: for example, given the name "TEST01", the file name "TEST01.SCH" will be internally used by the program.

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

The "Sheet General Information" dialogue box

In the "Sheet General Information" dialogue box, one can input the following data:


- *Title, sub-titles:* a title and three sub-titles that will be used as sheet captions;

- *Direction (°):* direction of the linear traverse along which the structural data have been collected;

- *Dip (°):* dip of the linear traverse along which the structural data have been collected;

- *Length (m):* length of the linear traverse along which the structural data have been collected.

The last three values are only used for the evaluation of the joint set frequencies, which are required for the calculation of the anisotropic permeability tensor of the rock mass.
Once these data have been entered, click the <Ok> button to close the window and open the Structural Data Collection Sheet dialogue box.

The "Structural Data Collection Sheet" dialogue box

In the "Structural Data Collection Sheet" dialogue box, one can input the following data:
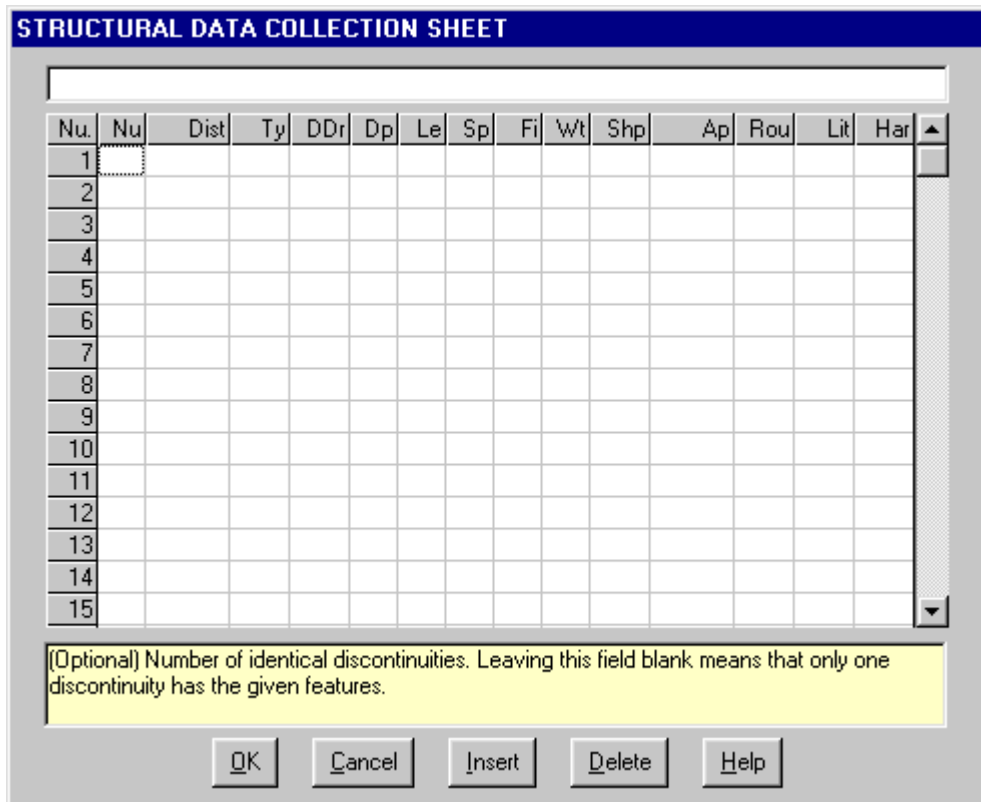
- **NU***mber:* if two or more discontinuities with exactly the same characteristics have been collected, it is possible to input the total number of identical discontinuities. This field is optional, and if left blank, a value of 1 is assumed.

- **DI***stance*: this is the distance of the discontinuity from the origin of the linear traverse. This field is optional.

- **TY***pe*: this is the type of the discontinuity. The following types are acknowledged: JN (joint), BD (bedding), CT (contact), FA (fault), SC (schistosity), SH (shear plane), VN (vein). This field is optional.

- **D***ip **DiR***ection*: this is the dip direction of the discontinuity (in degrees). This field is mandatory.

- **DiP**: this is the dip of the discontinuity (in degrees). This field is mandatory.

- **LE***ngth:* this is length class of the discontinuity. The following codes are acknowledged: A (<2 cm), B (2÷10 cm), C (10÷50 cm), D (50÷200 cm), E (200÷500 cm), F(>500 cm). This field is optional.

- **SP***acing*: the average space class of equal discontinuity groups. The following codes are acknowledged: A (<2 cm), B (2÷10 cm), C (10÷50 cm), D (50÷200 cm), E (200÷500 cm), F (>500 cm). Entering data in this field is necessary in case the number of equal discontinuities is more than 1. This field is optional.

- **FI***lling*: the type of filling found inside the discontinuity. The following codes are acknowledged: A = clay, B = breccia, C = calcite, D = iron oxides, E = sand, F = debris. This field is optional.

- **WA***ter*: water presence. The following codes are acknowledged: 2 = dry, 4 = humid, 5 = dripping, 6 = flow. This field is optional.

- **SH***ape:* shape of the discontinuity. The following codes are acknowledged: PLA = plane, SEG = segmented, WAV = wavy. This field is optional.

- *AP*erture: this is the aperture of the discontinuity (in millimetres). This field is optional; it is only used in the calculation of the anisotropic permeability tensor of the rock mass.

- *RO*ughness: roughness of the discontinuities surface. The following codes are acknowledged: LEV (levigated), SMT (smooth), ROU (rough), SEG (segmented), WAV (wavy). These data are optional.

- *LI*thology: The following codes are acknowledged:

   SHL = Clay - SLN = Siltstone - LMS = Limestone - SND = Sandstone - CGM = Conglomerate - DLM = Dolomite - TFF = Tuff - GBR = Gabbro - GRN = Granite - BSL = Basalt - GSS = Gneiss - AMI = Amphibolite - CSS = Calcar. Schist - QRZ = Quartzite - PRA = Green Schist - SRP = Serpentinites - SCS = Gen. Schist - SCA = Clay Schists - MLN = Mylonites - PHL = Phyllites - MBL = Marble - CTI = Cataclasite

- *HA*rdness: The following codes are acknowledged: HG (high), MD (medium), LW (low). These data are optional.

In the fields where these predefined codes are to be entered, the program will show the accepatble codes in the windows title bar, and will not accept entering other data.

Sheet Selection Command

This command enables one to select one or more sheets from which the structural data should be extracted to perform the cluster analysis.

The "Sheet Selection" dialogue box

All the structural data sheets that have previously been saved are listed in the "Sheet Selection" dialogue box. To select or de-select a sheet, click on the check box placed on the left of the sheet file name.
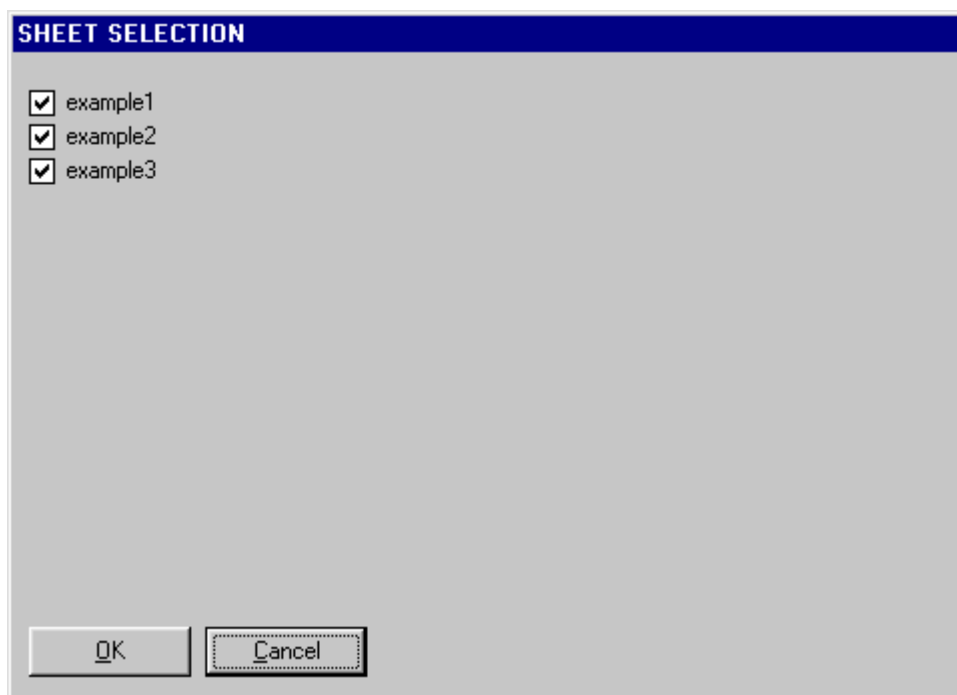
If more than 30 sheet files have been saved in the same directory, they will be paged in groups of 30 file names, and each time the <Ok> button is clicked, the "Sheet Selection" dialogue box will be shown again.
A maximum of 300 discontinuities can be used in the same cluster analysis.

Discontinuity Selection Command

This command allows one to define a criterion to extract a subset of discontinuities from the selected sheets.
This (optional) command is used, for example, to discard the faults and only extract the joints.
If no selection criterion has been defined, all the discontinuities in the chosen sheets will be used for the cluster analysis by default.

The "Discontinuity Selection" dialogue box

The "Discontinuity Selection" dialogue box contains a list of the quantities that have been entered into the "Structural Data Collection Sheet".
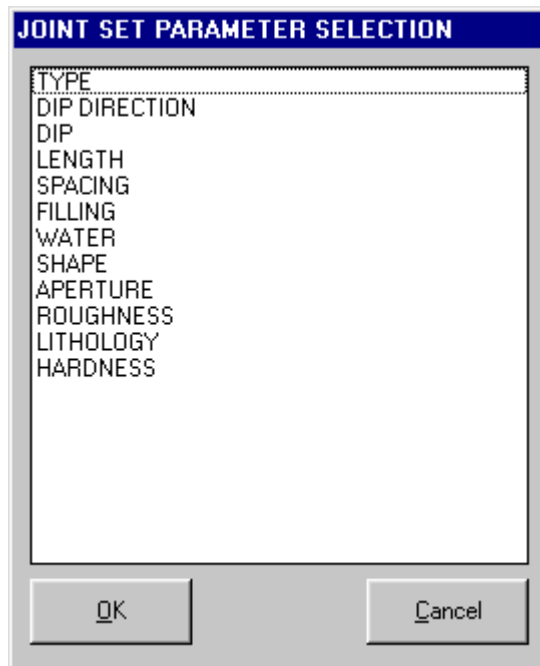
Select a quantity and click the <OK> button: a new window is opened.
All the acknowledged codes are selected by default. To de-select or select one of the available options for the chosen quantity, click on the check box placed on the left side of the window.

Slope Parameters Command

This command enables one to enter and edit a set of parameters related to the slope size and orientation, to the external forces, to the bolting of the slope and to the seismicity coefficients.

The "Slope Parameters" dialogue box

In the "Slope Parameters" dialogue box, one can input the following data:
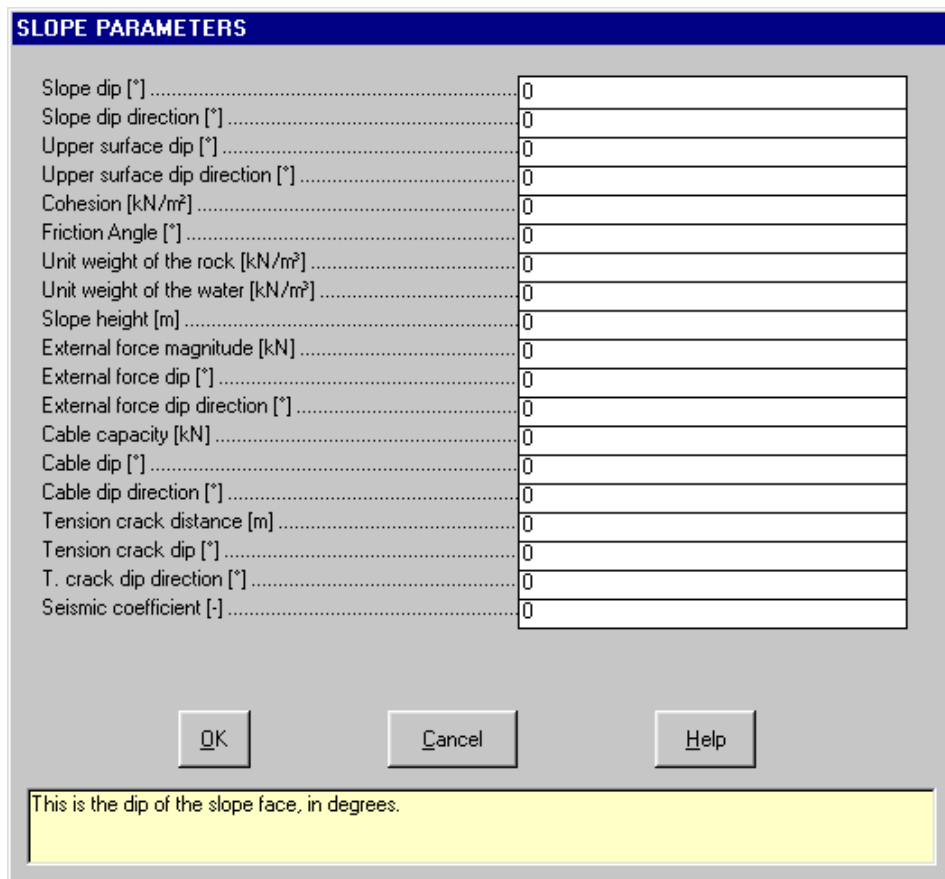
- *slope dip:* this is the dip of the slope face, in degrees.

- *slope dip direction:* this is the dip direction of the slope face, in degrees.

- *upper surface dip:* this is the dip of the upper surface of the slope, in degrees.

- *upper surface dip direction:* this is the dip direction of the upper surface of the slope, in degrees.

- *Cohesion:* this is the average cohesion of the discontinuities, in kN/m^2; this should be the value that is measured along the discontinuity surface and not the value of the hard rock. This value can be overwritten, after the cluster analysis, by assigning a cohesion value to each joint set individually.

- *friction angle:* this is the average friction angle of the discontinuities, in degrees; this should be the value that is measured along the discontinuity surface and not the value of the hard rock. This value can be overwritten, after the cluster analysis, by assigning a friction angle value to each joint set individually.

- *unit weight of the rock:* this is the average unit weight of the rock mass, in kN/m³.

- *unit weight of water:* this value can be used to evaluate the effects of the drainage on the stability conditions of the slope. A value of 0 kN/m³ simulates a completely dry slope, a value of 10 kN/m³ simulates a completely saturated slope, and values in the range 0-10 kN/m³ simulate a partially drained slope.

- *slope height:* this is the minimum elevation difference between the slope toe and the upper surface, in metres.

- *external force magnitude:* this is the magnitude of the resultant of the external forces applied to the rock wedge, in kN; if no external force is present, one can assign a zero value to this parameter.

- *external force dip:* this is the dip of the resultant of the external forces, in degrees.

- *external force dip direction:* this is the dip direction of the resultant of the external forces, in degrees.

- *cable capacity:* this is the capacity of a cable (if present) applied to the rock wedges, in kN; if no cable is present, one can assign a zero value to this parameter. Note that the *same* force is applied to *all* the wedges: for this reason one should repeat the stability analysis on different subsets of discontinuities to find the optimal force direction of each wedge.

- *cable dip:* this is the dip of the cable, in degrees.

- *cable dip direction:* this is the dip direction of the cable, in degrees.

- *tension crack distance:* this is the minimum distance between the tension crack (if present) and the slope crest, in metres. If no tension crack is present, one can assign a zero value to this parameter.

- *tension crack dip:* this is the dip of the tension crack, in degrees.

- *tension crack dip direction:* this is the dip direction of the tension crack, in degrees.

- *seismic coefficient:* this is the value of the seismic coefficient, which must range between 0 and 1. The seismic force, that is equal to the product of the wedge weight multiplied by the seismic coefficient, is applied in the worst direction: this direction is calculated as a function of the orientation and friction angle of the sliding plane(s).

The program thus calculates the safety factor of the potentially unstable wedges using the given parameters.

Joint Set Parameters Command

This command enables one to assign the cohesion and friction angle values to each joint set.
It is necessary to enter these values if the cohesion and friction angle have not been assigned to each discontinuity in the Structural Data Collection Sheet, otherwise they are automatically calculated as the average of the entered values.
This command can only be used if the cluster analysis already has been performed.

The "Joint Set Parameters" dialogue box

In the "Joint Set Parameters" dialogue box, the list of joint sets is shown. The progressive number of the joint sets refers to the output of the cluster analysis.
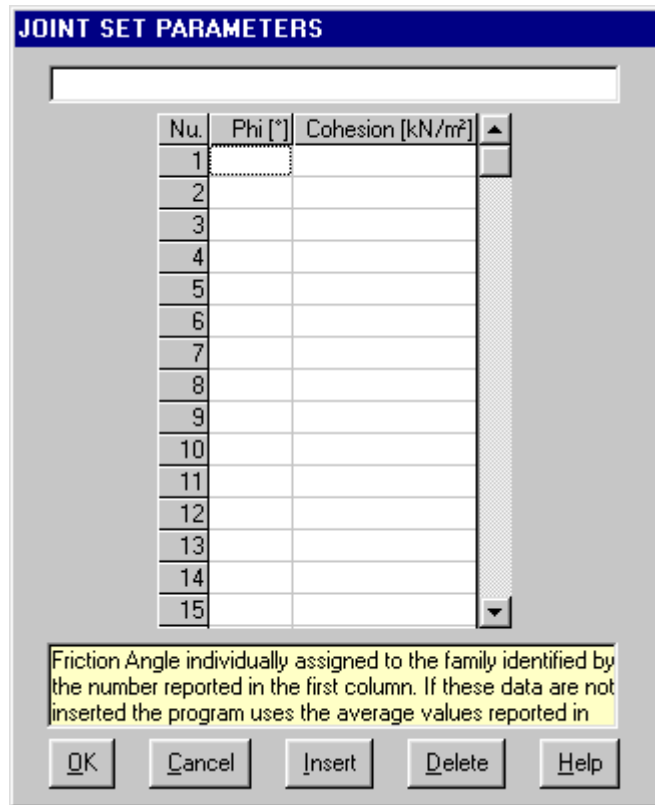
The cohesion and friction angle values, which are initially proposed by the program, have been automatically calculated as the average of the values entered in the Structural Data Collection Sheet.
The values entered or modified in this dialogue box are used for the rock wedge stability analysis.

Titles Command

This command enables one to enter and edit a title and three sub-titles. These titles are printed on the graphical and tabular outputs.
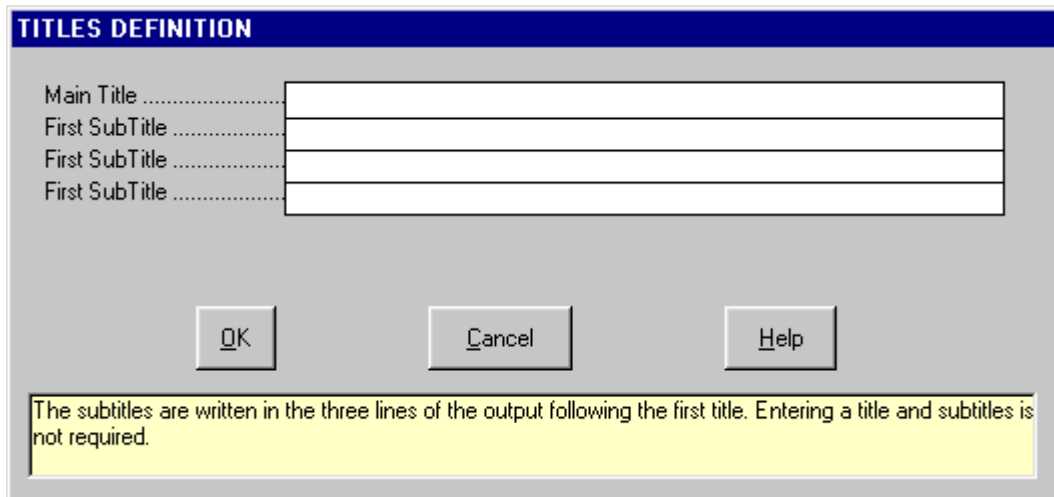
Calculate Menu

This menu is used to execute the cluster and the rock wedge stability analysis.
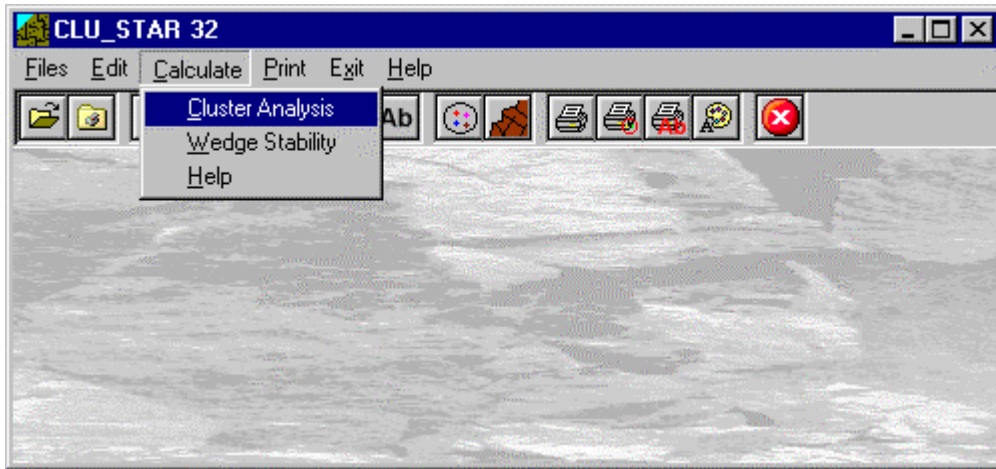
Cluster Analysis Command

The general principles of the hierarchical and non-hierarchical cluster analysis are outlined in the "Introduction to the program".

This command enables one to perform the cluster analysis according to the following procedure.

The stereographic projection of the discontinuities is drawn, and the poles are shown as circles with larger diameters if discontinuities with the same orientation are present.

A dialogue window is shown, which allows one to select one of the methods to calculate the distance between two clusters.
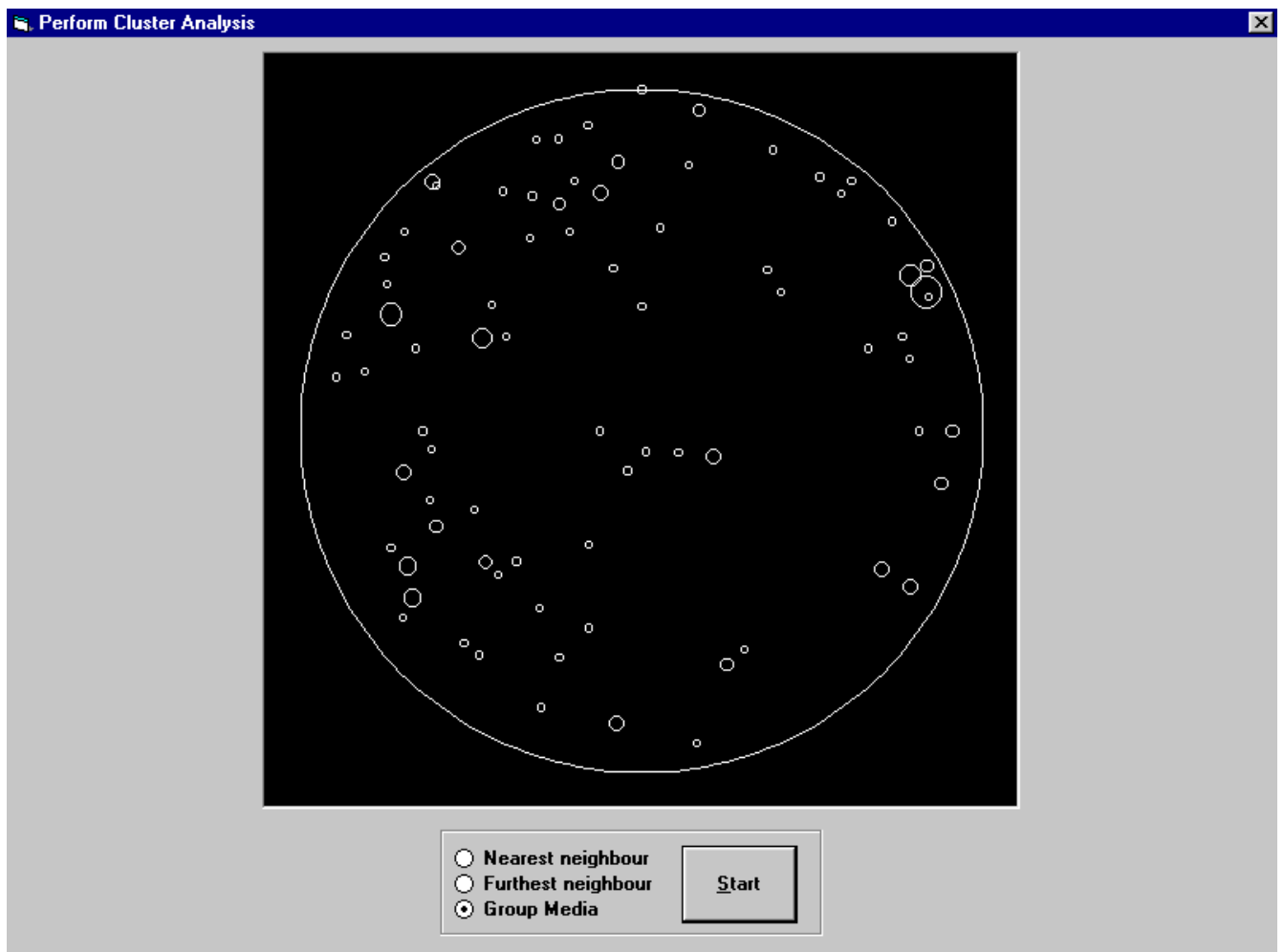
Each discontinuity is initially assigned to a single cluster. The program then performs a sequence of clustering steps by aggregating the two nearest clusters at each step. The clustering tree is completed when a single, large cluster is obtained that contains all the discontinuities.

A dialogue box is opened and a clustering level can be selected; once the initial clustering level has been entered, the tree can be scanned with the <Increase> and <Decrease> arrows.

When the optimal clustering level has been chosen, the limit angle value for the non-hierarchical clustering stage can be entered. The limit angle is an optional value and if one leaves the input field blank, the non-hierarchical clustering will not be performed.

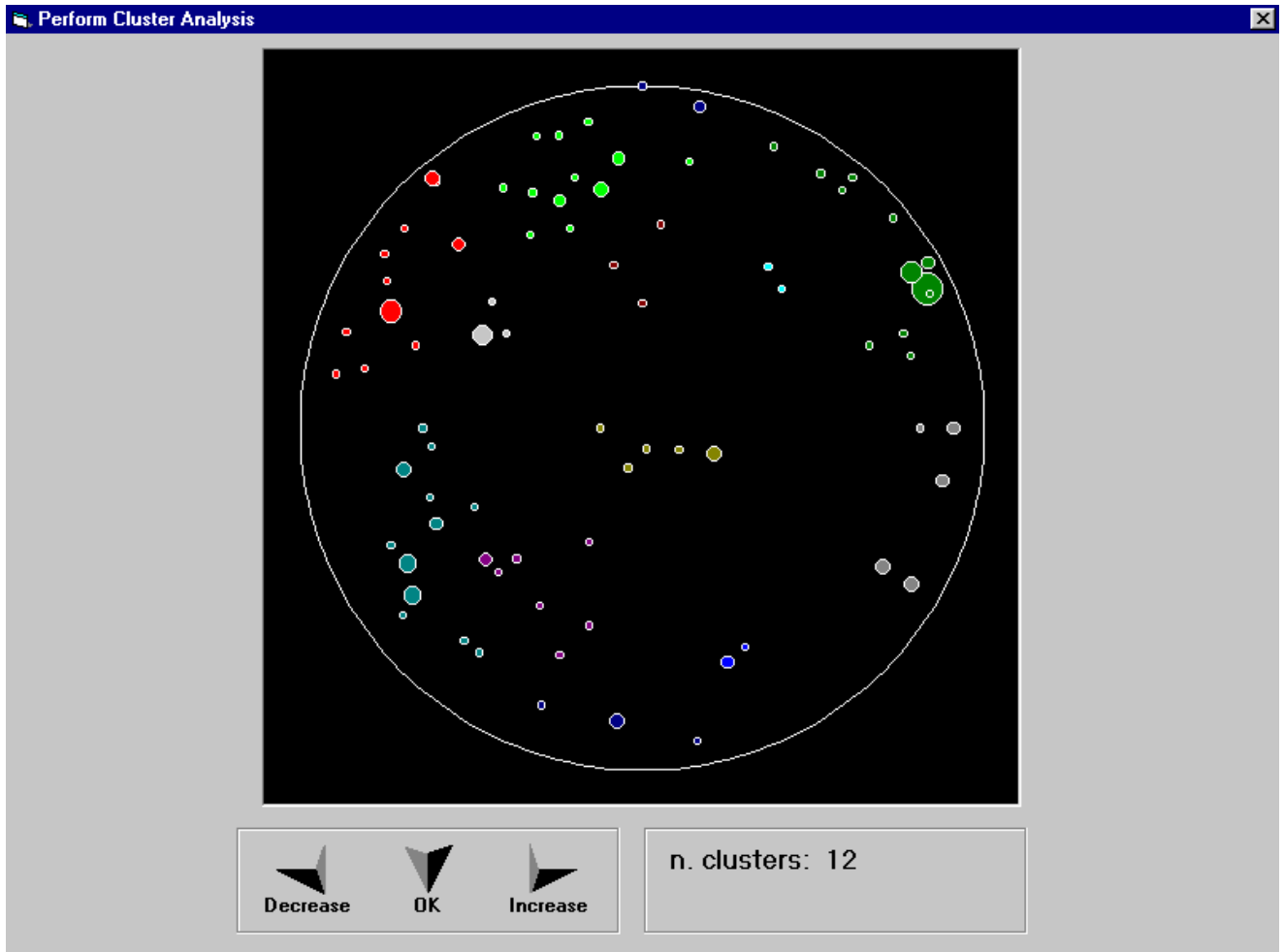As mentioned in the first chapter, one assumes, in the wedge stability analysis, that there are two discontinuities whose intersection passes through the toe of the slope, and this can only occur if a sufficiently large number of discontinuities belongs to the two joint sets.

The next stage consists in the selection of the larger joint sets that are used in the stability calculations. One can add or remove the joint sets, ordered according to the quantity of the contained discontinuities, using the <Increase> and <Decrease> arrows.

A dialogue box is then shown that allows one to exit the clustering stage or to return to the selection of the initial clustering level.

If the slope parameters have already been entered, the wedge stability analysis is performed.

Wedge Stability Command

This command enables one to perform or repeat the wedge stability analysis when the slope parameters have been entered or modified after the cluster analysis stage. For example, this allows one to verify, without repeating the full clustering procedure, whether the modification of the slope orientation can contribute to increase the safety of an excavation.

Print Menu

The Print menu is used to configure the graphical output, to plot the stereonets and the rose diagrams and to produce the listings of the calculation results.
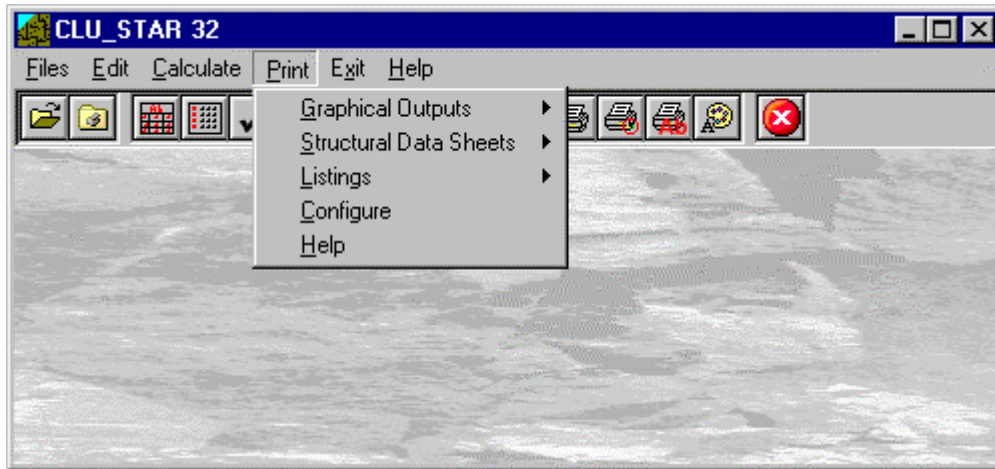
Graphical Output Command

This command allows one to select which kind of graphical output should be plotted.

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.

- **Print**: click the print icon to send the drawing to the default printer.

- **Resized print**: click this icon to send the drawing to the default printer. The image will be resized to fit the current paper size.

- **DXF**: one can export the graphic output as a DXF file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the DXF file, which has, by default, the same name as the current project.

- **EMF/W**: This icon allows the graphic output to be exported to a version of the Enhanced Windows Metafile which is compatible with Microsoft Word. Clicking this icon gives access to a dialogue window that enables one to assign a different name to the EMF file, which has, by default, the same name as the current project.

- **EMF/D**: This icon allows the graphic output to be exported to a version of the Enhanced Windows Metafile which is compatible with Corel Draw. Clicking this icon gives access to a dialogue window that enables one to assign a different name to the EMF file, which has, by default, the same name as the current project.

- **BMP**: one can export the graphic output as a BMP file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the BMP file, which has, by default, the same name

as the current project.

- **GIF**: one can export the graphic output as a GIF file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the GIF file, which has, by default, the same name as the current project.

- **JPG**: one can export the graphic output as a JPG file by clicking this icon; it gives access to a dialogue window that enables one to assign a different name to the JPG file, which has, by default, the same name as the current project.
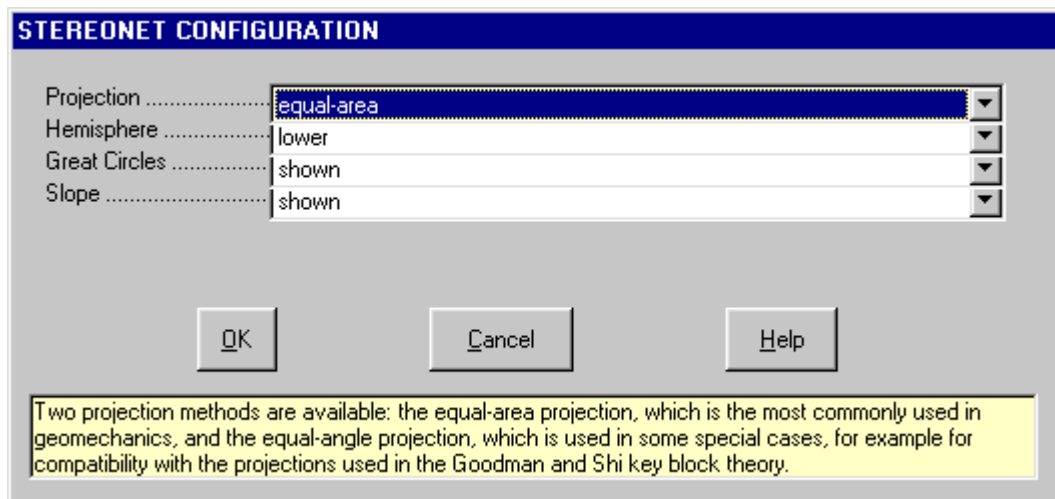
- **Exit**: click this icon to close the preview window and return to the main menu.

Un-clustered Joint Projection Command

The "Un-clustered Joint Projection Command" is used to plot the orientation of the collected discontinuities on a stereonet, disregarding the results of the clustering stage. It is therefore a simple graphical representation of the data collected in the structural geology data sheets.

The "Stereonet Configuration" dialogue box - Un-clustered Joint Projection

The program allows one to configure the projections plotted on the stereonets. The following data can be entered into the "Stereonet Configuration" dialogue box:

- *Projection:* two projection methods are available: the equal-area projection, which is the most commonly used in geomechanics, and the equal-angle projection, which is used in some special cases, for example for compatibility with the projections used in the Goodman and Shi key block theory.

- *Hemisphere:* poles and great circles can be projected onto the lower or upper hemispheres, for the same reasons explained in the previous parameter.

- *Great Circles:* these are the projections of the intersections of the planes with the reference sphere and can be plotted in addition to the pole projections.

- *Slope and Friction Angle:* the great circle that represents the slope face can be added to the plot, together with the friction angle circle, to allow one to perform the Markland test. This test is a little obsolete when compared to the analytical solution used by the program to calculate the wedge safety factors.

Joint Set Projection Command (*ALT,P,G,J*)

The "Joint Set Projection Command" is used to plot the orientation of the joint sets and collected discontinuities that result from the clustering stage on a stereonet. The use of different colours and symbols allows one to identify the joint sets both in colour and black and white printouts.

The "Stereonet Configuration" dialogue box - Joint Set Projection

The program allows one to configure the projections plotted on the stereonets. The following data can be entered into the "Stereonet Configuration" dialogue box:
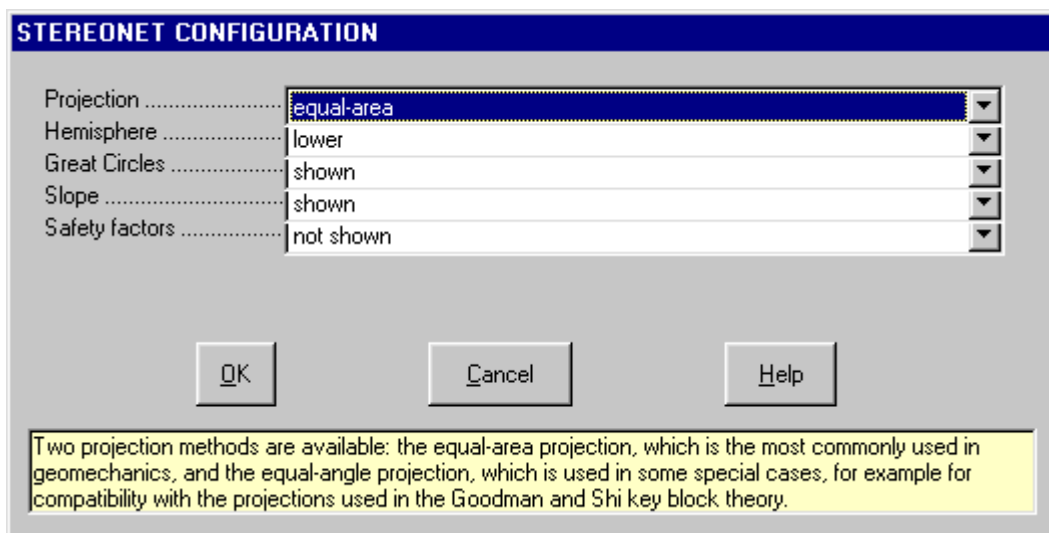
- *Projection:* two projection methods are available: the equal-area projection, which is the most commonly used in geomechanics, and the equal-angle projection, which is used in some special cases, for example for compatibility with the projections used in the Goodman and Shi key block theory.

- *Hemisphere:* poles and great circles can be projected onto the lower or upper hemispheres, for the same reasons explained in the previous parameter.

- *Great Circles:* these are the projections of the intersections of the planes with the reference sphere and can be plotted in addition to the pole projections.

- *Slope and Friction Angle:* the great circle that represents the slope face can be added to the plot, together with the friction angle circle, to allow one to perform the Markland test. This test is a little obsolete when compared to the analytical solution used by the program to calculate the wedge safety factors.

- *Safety Factors:* this option allows one to add a line in the direction of the intersection of the two planes and the safety factor value, for each potentially unstable wedge.

Rose Diagram Command

This command allows one to draw the collected structural data as a rose diagram.

Structural Data Sheets Command

The Structural Data Sheets Command allows one to print or export the listings of the collected structural data sheets. An Open dialogue box can be used to choose the sheet to be printed.

Sheet Preview Command

With this command, it is possible to preview the listings of the calculation results. The listings cannot be changed in the preview window.
The scroll bar, the UP arrow and DOWN arrow keys, or the PAGE UP e PAGE DOWN keys can be used to scroll the page. To exit press ESC or double-click any point of the window.

Sheet Direct Print Command

With this command, it is possible to directly send the listings of the calculation results to the printer.

Sheet Export to DOC

With this command, it is possible to export the listings of the calculation results into the DOC format (Microsoft Word 97 compatible).
The program creates a file with the same name as the current project but with the ".DOC" extension; the file is saved in the same directory as the data files.
The file is automatically opened if the program finds the Microsoft Word executable.

Sheet Export to TXT

With this command, it is possible to export the listings of the calculation results into the TXT format (Microsoft Notepad compatible).

The program creates a file, with the same name as the current project but with the ".TXT" extension; the file is be saved in the same directory as the data files.
The file is automatically opened if the program finds the Microsoft Notepad executable.

Sheet Export to SLK

With this command, it is possible to export the listings of the calculation results into the SLK format (Microsoft Excel 97 compatible).
The program creates a file, with the same name as the current project but with the ".SLK" extension; the file is saved in the same directory as the data files.
The file is automatically opened if the program finds the Microsoft Excel executable.

Listings Command

This command allows one to print or export the listings of the cluster analysis results.

Listing Preview Command

With this command, it is possible to preview the listings of the calculation results. The listings cannot be changed in the preview window.
The scroll bar, the UP arrow and DOWN arrow keys, or the PAGE UP e PAGE DOWN keys can be used to scroll the page. To exit press ESC or double-click any point of the window.

Listing Direct Print Command

With this command, it is possible to directly send the listings of the calculation results to the printer.

Listing Export to DOC

With this command, it is possible to export the listings of the calculation results into the DOC format (Microsoft Word 97 compatible).
The program creates a file with the same name as the current project but with the ".DOC" extension; the file is saved in the same directory as the data files.
The file is automatically opened if the program finds the Microsoft Word executable.

Listing Export to TXT

With this command, it is possible to export the listings of the calculation results into the TXT format (Microsoft Notepad compatible).
The program creates a file, with the same name as the current project but with the ".TXT" extension; the file is be saved in the same directory as the data files.
The file is automatically opened if the program finds the Microsoft Notepad executable.

Listing Export to SLK

With this command, it is possible to export the listings of the calculation results into the SLK format (Microsoft Excel 97 compatible).
The program creates a file, with the same name as the current project but with the ".SLK" extension; the file is saved in the same directory as the data files.
The file is automatically opened if the program finds the Microsoft Excel executable.

Configure Command

With this command, one can define the graphical properties of the different objects in the graphical output.
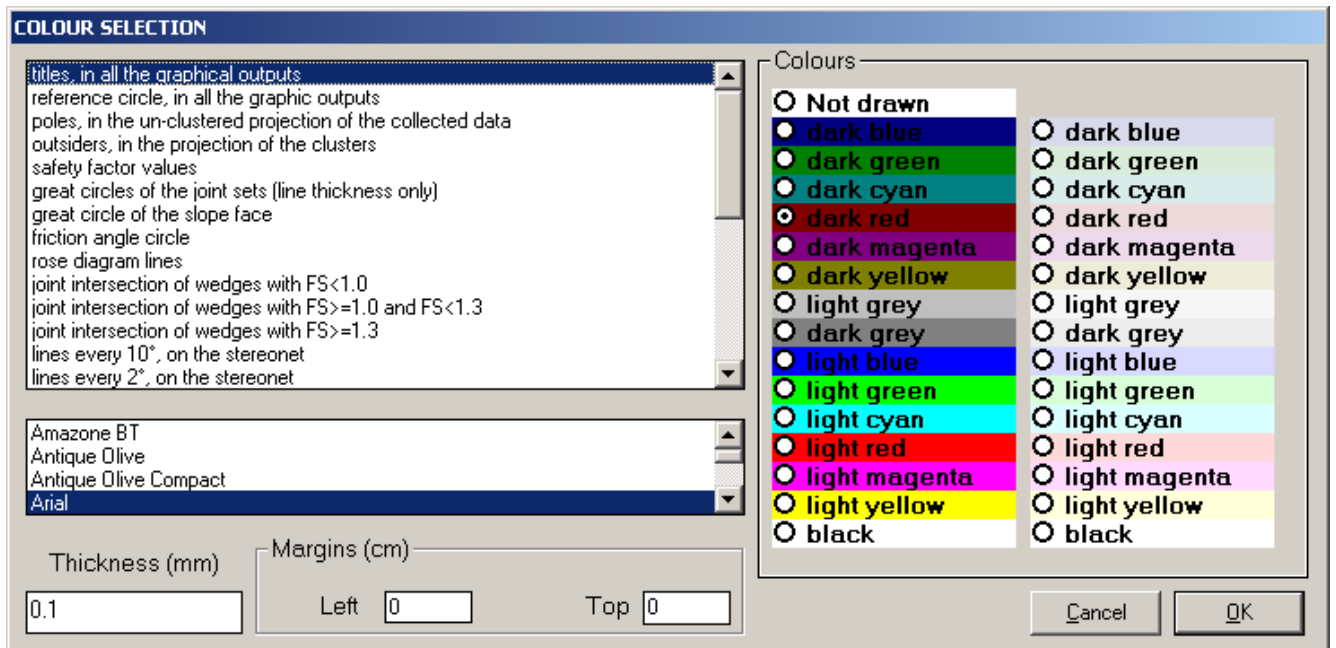
One can associate a colour, a line thickness and a character font to each object. Notice that not all the objects have both a line thickness and a character font: for example, a *title* only requires the colour and the character font to be defined.

- 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.