

# Site Data Management

---

Version 8.095



Site – Site Data Management – Version 8.095  
User Guide

Warning: The Site module and the SM2 library and this manual are protected by copyright laws and by international conventions. Any reproduction or distribution of the program or its manual, partially or totally, by any means, is strictly forbidden, unless a written permission is given by SOBEK TECHNOLOGIES INC. Any person who does not respect these rules is guilty of a misdemeanour of infringement and is punishable by law.

Published by: SOBEK TECHNOLOGIES INC.  
4205 Northcliffe  
Montreal (Quebec)  
H4A 3L2  
Tel: 514 285-4873  
Email: [genevieve@sobek-technologies.com](mailto:genevieve@sobek-technologies.com)

The information in this manual can be modified without notice and would not be binding to SOBEK TECHNOLOGIES INC. **This version of the documentation was updated in September 2019 with version 8.095 of Geotec.**

#### TRADEMARKS:

In this guide, we refer to these registered products:

Access is a registered trademark of Microsoft Corporation  
Acrobat PDFWriter is a registered trademark of Adobe Systems Incorporated  
AutoCAD is a registered trademark of Autodesk, Inc.  
ORACLE is a registered trademark of Oracle Corporation  
SQL Server is a registered trademark of Microsoft Corporation  
Windows is a registered trademark of Microsoft Corporation  
XVT Portability Toolkit is a registered trademark of XVT Software, Inc.

---

## SITE - TABLE OF CONTENTS

---

<b>LIST OF FIGURES</b>	<b>iv</b>
<b>Chapter 1. Site - Site Data Management</b>	<b>1-1</b>
1. Launching the Site module	1-1
2. File, Edit, Insert, Window and Help menus	1-1
3. View menu	1-1
4. Tools menu	1-2
4.1. Options	1-2
4.1.1. Maps and domains display	1-2
4.1.2. Geotechnical data display	1-4
5. Data menu	1-5
6. Organization of a graphic output	1-5
7. Overview of graphs	1-6
8. The primary graph	1-8
8.1. The PLAN_VIEW graph	1-8
8.1.1. Predefined curves	1-8
8.1.2. Curves of information	1-10
8.2. The 3D_VIEW	1-11
9. The sub-graphs	1-12
9.1. Creation of a property graph	1-12
9.2. Types of curves in the sub-graphs	1-13
9.3. Format of the numerical values	1-13
9.4. HEADER sub-graph	1-13
9.5. STRATIGRAPHY sub-graphs	1-14
9.5.1. Layers and sub-layers	1-14
9.5.2. Display of the STRATIGRAPHY sub-graph for the CODE1 field	1-16
9.5.3. Display of the stratigraphic symbols	1-16
9.6. SAMPLE sub-graphs	1-17
9.6.1. Display of the SAMPLE_NO curve	1-17
9.6.2. Display of the STATE curve	1-17
9.6.3. Display of the USCS curve	1-18
9.6.4. Display of the MATERIAL curve	1-18
9.6.5. Selection of the types of samples to present	1-18
9.7. PIEZOMETER sub-graph	1-18
9.8. LIMITS sub-graph	1-19
9.9. CONSOLIDATION sub-graph	1-19
9.10. ROCK_JOINT sub-graphs	1-20
9.11. CPTU_ZONE sub-graphs	1-20
9.12. Display of property values or of concentrations of contaminants	1-20
9.13. Display of elevations and information associated to a depth curve	1-21
9.14. Display of equivalences to values	1-22

10.	Interactive editing	1-22
10.1.	Interactive editing in 3D view	1-22
10.2.	Interactive editing in 2D view	1-22
10.2.1.	Creating a site	1-23
10.2.2.	Creating an axis	1-23
10.2.3.	Modifying an axis	1-23
10.2.4.	Creating borings	1-24
10.2.5.	Modifying a boring	1-25
10.2.6.	Selecting borings	1-25
10.2.7.	Creating and modifying a rectangular limit	1-26
10.2.8.	Creating and modifying a polygonal limit	1-26
<b>Chapter 2.</b>	<b>SM2 - Site Mapping and Modeling</b>	<b>2-1</b>
1.	Launching the SM2 library	2-1
2.	File, Edit, Insert, Window and Help menus	2-1
3.	View menu	2-1
4.	Tools menu	2-2
5.	Organization of a graphic output	2-2
6.	Mapping menu	2-2
6.1.	Properties and editing of a cartographic entity	2-3
6.1.1.	Name of layer	2-3
6.1.2.	Type of entity	2-3
6.1.3.	Elevation of the points	2-3
6.1.4.	Closing a polyline	2-4
6.1.5.	Drawing attributes of the entity	2-4
6.1.6.	Table of points	2-5
6.2.	Creating a cartographic entity	2-5
6.3.	Copying a cartographic entity	2-6
6.4.	Moving a cartographic entity	2-6
6.5.	Linking two cartographic entities	2-6
6.6.	Splitting a cartographic entity	2-6
6.7.	Deleting a cartographic entity	2-7
6.8.	Adding a point	2-7
6.9.	Moving a point	2-7
6.10.	Deleting a point	2-7
6.11.	Pop-up menu	2-7
7.	Modeling menu	2-8
7.1.	Map management	2-8
7.1.1.	Data file	2-8
7.1.2.	Import of a data file	2-8
7.1.3.	Background file	2-9
7.1.4.	File of polygonal limit	2-9
7.1.5.	File domain and additional information	2-9
7.1.6.	Operations on the file domain	2-10
7.1.7.	Data domain	2-11
7.1.8.	Other operations	2-11
7.2.	Layer editing	2-11
7.3.	Recursive kriging	2-13
7.4.	Linear Triangulation	2-16
7.5.	Data Terrain Model	2-18
7.5.1.	Calculate the DTM from a kriged file	2-18
7.5.2.	Calculate the DTM from a triangulated file	2-20

---

7.5.3.	Horizontal plane (domain S1)	2-21
7.5.4.	Add a constant ( $S1 + Z$ )	2-22
7.5.5.	Subtract a constant ( $S1 - Z$ )	2-23
7.5.6.	Add two surfaces ( $S1 + S2$ )	2-23
7.5.7.	Subtract two surfaces ( $S1 - S2$ )	2-23
7.5.8.	Upper surface ( $S1, S2$ )	2-23
7.5.9.	Lower surface ( $S1, S2$ )	2-24
7.5.10.	Intersection of two surfaces ( $S1, S2$ )	2-24
7.5.11.	Calculate Z from a given surface ( $S1$ )	2-24
7.6.	Isovalue Contours	2-25
7.7.	Volumes	2-27
7.7.1.	Volumes between two surfaces	2-27
7.7.2.	Volumes by elevations	2-29
7.8.	Color Selection	2-31
7.8.1.	Color file	2-31
7.8.2.	Color selection by elevation	2-32
7.8.3.	Application of colors	2-32
7.8.4.	Other parameters	2-33

## LIST OF FIGURES

Figure 1-1 - Data display options	1-3
Figure 1-2 - Graph editing window	1-6
Figure 1-3 - Predefined curves of the PLAN_VIEW graph	1-10
Figure 1-4 - Example of textual information added in PLAN_VIEW	1-11
Figure 1-5 - Predefined curves of a STRATIGRAPHY sub-graph	1-15
Figure 1-6 - Symbols representing the state of the samples	1-18
Figure 1-7 - Interactive menu in 3D view	1-22
Figure 1-8 - Interactive menu in 2D view	1-23
Figure 1-9 - Editing an axis	1-24
Figure 1-10 - Creating new borings	1-24
Figure 1-11 - Editing a boring	1-25
Figure 1-12 - Selection of borings	1-26
Figure 2-1 - Mapping menu	2-2
Figure 2-2 - Example of a highlighted entity and its properties	2-3
Figure 2-3 - Property window of an entity	2-4
Figure 2-4 - Modeling menu	2-8
Figure 2-5 - Map editing window	2-10
Figure 2-6 - Layer editing window	2-12
Figure 2-7 - Calculation by kriging	2-14
Figure 2-8 - Plan view during kriging	2-15
Figure 2-9 - Calculation by triangulation	2-17
Figure 2-10 - Plan view before and after triangulation	2-18
Figure 2-11 - List of operations on the DTM	2-18
Figure 2-12 - Calculate the DTM from a kriged file	2-19
Figure 2-13 - Example of a calculated DTM in 3D and in OpenGL mode	2-20
Figure 2-14 - Create a horizontal plane (domain S1)	2-21
Figure 2-15 - Add a constant (S1 + Z)	2-22
Figure 2-16 - Window for the calculation of contour lines	2-26
Figure 2-17 - Example of an isovalue contours file in 3D	2-27
Figure 2-18 - Calculation of volumes between two surfaces	2-28
Figure 2-19 - Calculation report of volumes between two surfaces	2-29
Figure 2-20 - Calculation of volumes by elevations	2-30
Figure 2-21 - Calculation report of volumes by elevations	2-31
Figure 2-22 - Color Preferences window	2-32

## CHAPTER 1. SITE - SITE DATA MANAGEMENT

The **Site** module is used for the interactive treatment of geographical information relative to sites. It allows the user to display borings and their geotechnical properties in a spatial system in 2D and 3D in addition to a cartographic map.

Site uses a Graphical User Interface (i.e. with windowing) that is compatible with Microsoft Windows 7, 8 and 10. Site uses the Geo, X3D and SM2 libraries created by Sobek Technologies Inc. for data editing, the definition of the presentation style, and viewing and editing graphic results. Please refer to the guides for Geo, X3D and SM2 (chapter 2 of this documentation) for all details.

---

### 1. LAUNCHING THE SITE MODULE



Double-click on this icon to open the **Site** module.

The connection to the database can be done directly when calling Site by entering the following string as target. The string will be:

***C:\Geotec80\Bin\Site.exe username/password@odbc:connection\_alias***

See paragraph 1 of chapter 2 of the X3D guide for all the details about the connection to a database. Once the connection to a database is done, the user can execute a data query to produce a plan or 3D view.

When **Site** is first launched, the following files are loaded from **C:\Geotec80\Style**:

- the markers file **geotec8.mrk**;
- the patterns file **geotec8.ptn**;
- the style file **site\_landscape.sty** which is a classic presentation model with a plan view.

The user can begin working with this initial presentation, or open another style file included with the software. He can modify the style file and save it in a new presentation. In subsequent loadings, Site opens the style, markers and patterns files that were in use when Site was closed.

---

### 2. FILE, EDIT, INSERT, WINDOW AND HELP MENUS

None of the options of the **File**, **Edit**, **Insert**, **Window** and **Help** menus is specific to Site. The detailed description of each option of these menus is done in chapters 2, 3, 4 and 8 of the X3D guide.

---

### 3. VIEW MENU



The **2D View** and **3D View** options of the **View** menu and this button are used to change the view of the PLAN\_VIEW graph. In the axes editing window, it is possible to change from 2D to 3D view and vice-versa for PLAN\_VIEW. For the sub-graphs, the box cannot be checked.



The **OpenGL mode** is used to view solids and surfaces with hidden lines and faces and shadows in 3D. The **XVT mode** is the standard display mode.

The last column (OGL) of the graph editing window (see Figure 1-2) is used only for the OpenGL mode in 3D. The user enters the value of the radius corresponding to the maximum scale for the numerical values. For example, if an OGL of 5 is entered for the SAMPLE graph displaying the RQD of the samples, an RQD of 100% will be 5m in OpenGL, of 20% will be 1m, etc.

In OpenGL mode, the user can then save his data file in an .igs format to export in CATIA.

## 4. TOOLS MENU

The options of the **Tools** menu are used to define the utilisation parameters of Site and to edit the graphic components produced by Site: the characteristics of the graphic page, the graph, sub-graphs and their curves, the legends, the axes, the markers and patterns used. All these options are described in details in chapter 6 of the X3D guide.

**Options** is specific to Site. It is presented in paragraph 4.1.



The **SM2 Mode** is also specific to Site. It opens the SM2 library for the site mapping and modeling. See chapter 2 for all the details.

### 4.1. Options



The **Options** item of the **Tools** menu or this button opens the window of options for data display, shown in Figure 1-1, where the user can define the display parameters of the borings, axes, limits, etc. in the plan view (and 3D view).

#### 4.1.1. Maps and domains display

- Upon navigating, the user can choose to display only the **current site** or **all the sites** at once. When all the sites of a query are displayed, they are shown according to the option selected for the **calculated domain** (see below). By navigating among the records, the current site is shown in red.
- The user can **display** the **site map** and/or the **region map** in the plan view or 3D view by checking the corresponding box. The site map is entered in the **FILE\_NAME** field of the **SITE** table, must be located in the directory of the input files and of a **.sit extension**. The region map is entered in the **FILE\_NAME** field of the **REGION** table, for the region associated to the site and must also be in the directory of the input files or have its complete path and be of **.sit extension**.
- A scrolling list is used to choose the **basis of the calculated domain**:
  - **The SITE table** - based on the EASTING, NORTHING, WIDTH, HEIGHT and ANGLE fields of the SITE table.
  - **The site map** - based on the site map in a .sit file (the **site map** must be displayed).
  - **The REGION table** - based on the EASTING\_MIN, EASTING\_MAX, NORTHING\_MIN and NORTHING\_MAX of the REGION table for the region associated to the site.



- **The region map** - based on the region map in a .sit file for the region associated to the site (the **region map** must be displayed).
- **Borings location** - the area including all the borings retrieved. This mode is particularly useful for the early detection of errors in the coordinates.
- **Axes location** - the area including all the turning points of the axes and the associated profiles retrieved (for the 3D view).
- **Axes and borings location** - the area including all the borings and axes retrieved.
- **Limit defined domain** - the area including the polygonal limit retrieved, the one whose name is the same as the site retrieved, or the one that has the site retrieved as reference.
- **All available data** - the area including all borings, axes and domains associated to the sites and regions and to their respective topographic maps.
- Check the box **Display domains** to display the outlines of the domains specified below.
- Check the box **Detect domains** to display information about the domains in a tooltip when the user moves the mouse on a domain outline. The domains must be displayed for this option to be available.
- With the option **Display file with extension**, the user can display an additional topographic file, on top of the site map (.sit). The file displayed will be the one whose name is indicated in the FILE\_NAME field of the SITE table, located in the input files directory, and with the extension selected in the scrolling list.

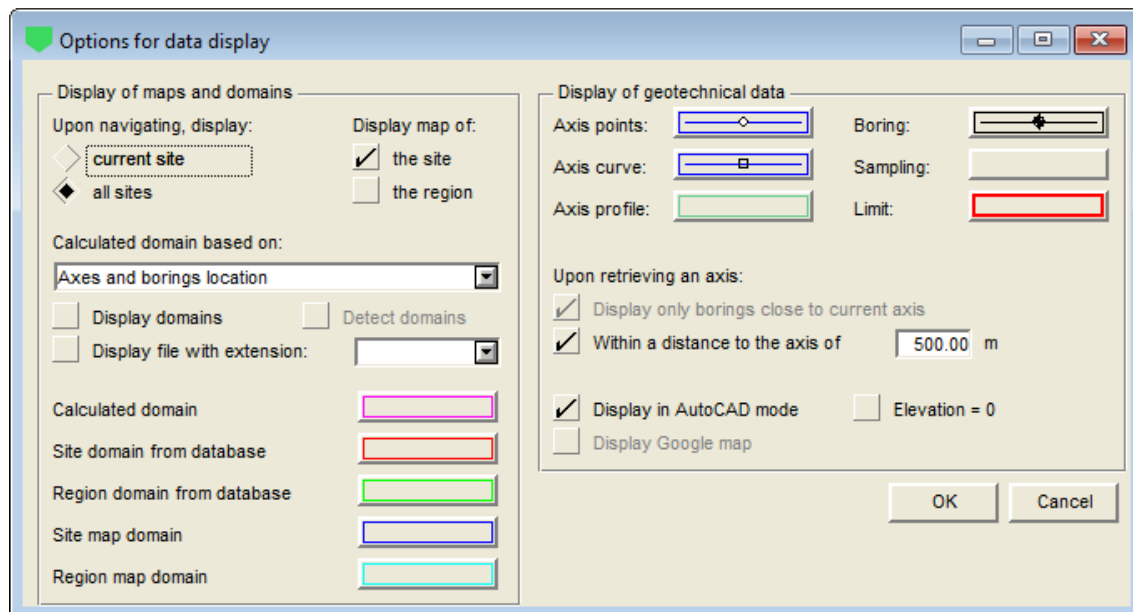


Figure 1-1 - Data display options

- The next five buttons open the drawing attributes configuration window, and correspond to the curves numbered #130 to #134 in the legend and curves editing window for the PLAN\_VIEW graph (see figure 1-3). It is possible to define the attributes from both locations. See paragraph 8.1 for all the details.

- The button next to the label **Calculated domain** corresponds to the **dom\_calculated** curve (#134), which delimits the resulting domain, calculated according to the option selected in the drop-down list.
- The buttons next to the labels **Site domain from database** and **Region domain from database** correspond to the **dom\_site\_db** (#130) and **dom\_region\_db** (#131) curves, which delimit the domain of the site and of the region, as defined in the database.
- The buttons next to the labels **Site map domain** and **Region map domain** correspond to the **dom\_site\_map** (#132) and **dom\_region\_map** (#133) curves, which delimit the site map and region map in the .sit files.

#### 4.1.2. Geotechnical data display






- The buttons in this section open the drawing attributes configuration window, and correspond to the curves numbered 100 to 102, 110 and 120 in the legend and curves editing window for the PLAN\_VIEW graph (see figure 1-3). It is possible to define the attributes from both locations. See paragraph 8.1 for all the details.
  - The **Axis points** button corresponds to the **Axis\_Points** curve (#100) and is used to define the line passing through the turning points of the axes and the markers at these points.
  - The **Axis curve** button corresponds to the **Axis\_Curve** curve (#101) and is used to define the curvature of the axes, their identification and their symbol for their end points.
  - The **Axis profile** button corresponds to the **Axis\_Profile** curve (#102) and is used to define the position of the geophones and the profiles obtained.
  - The **Boring** button corresponds to the **Boring** curve (#110) and is used to define the identification of the borings in the plan view, their symbol and their projection.
  - The **Sampling** button is not used.
  - The **Limit** button corresponds to the **Limit** curve (#120) and is used to define the rectangular or polygonal limits.

If the retrieval is done via the Axis entry form (similarly to the usage in Pro):

- Check the box **“Display only borings close to current axis”** to retrieve a rectangular limit around the axis, of a distance specified in the following **field (m)**. If the option ‘display all sites’ is selected, Site retrieves only the borings around the current axis, otherwise a very large number of data could be returned. In this case, the box cannot be unchecked.
- Check the box **“Within a distance to the axis of”** to display only the borings located at a distance to the axis smaller or equal to the **distance (m)** indicated in the related field.
- Check the box **Display in AutoCAD mode** to set the texts in geographic coordinates instead of millimeters, for a better compatibility of the .dxf files produced for AutoCAD. This box is not active if the Google map is displayed.
- The **Elevation = 0** checkbox is used solely for AutoCAD mode. The distances are then calculated horizontally instead of in 3 dimensions, using elevations of 0.
- Check the box **Display Google map** to display the Google map in background. This box is not active if the AutoCAD mode is used.

## 5. DATA MENU

The options of the **Data** menu are detailed in chapter 7 of the X3D guide. The options are mainly used to retrieve data in the tables of the **Geotec** database. In Site, the default retrieval (**simple query**) is done on the **Site** table or the **Axis** table.

In Site, the data entry options available in the **Data** menu (**Entry** option) and with the buttons of the application toolbar (    ) are those of **Site**, **Axis**, **Boring**, **Stratigraphy** and **Sample**. The user first inserts the sites, followed by the axes and borings related to the sites, then the stratigraphy of the borings, and finally the samples from the borings.



The **Color Selection** option is used for the cartographic maps in OpenGL mode. It is detailed in chapter 2.

## 6. ORGANIZATION OF A GRAPHIC OUTPUT



The definition of graphs is established and updated in the graph editing window opened by selecting the **Graphs...** option of the **Tools** menu or by clicking this button. More details about the graph editing are available in paragraph 3 of chapter 6 of the X3D guide. Figure 1-2 shows the graph editing window in Site.

The table in this window contains all the graphs defined in the style file used. Each graph is identified by a unique number in the **#** column. The first graph is always **Page** with **# 0**. This graph represents the graphic page and cannot be deleted.

Among the graphs, there is the **primary graph PLAN\_VIEW** whose **#P cell is 0** and the **sub-graphs** whose **#P cell is a negative value**.

Site produces one type of **primary graph**: a plan view (**PLAN\_VIEW**), which is graph #1. This graph cannot be deleted. This graph can also be put in 3 dimensions. Use the View menu to set the graph in 2D or 3D. A topographic background can be displayed in the plan view (in 2D and 3D).

The plan view shows the location of the borings with markers. Specific information from the BORING table can also be displayed under each boring point displayed. In addition, the axes can be displayed, markers showing their turning points and their ends. Paragraph 8.1 presents this graph.

When the graph is in 3D, the tridimensional view shows as columns the borings and their properties defined as sub-graphs. Paragraph 8.2 presents this graph.

For the primary graph, the **Left**, **Width**, **Right**, **Bottom**, **Height** and **Top** fields define its position and dimensions; the user can modify these values.

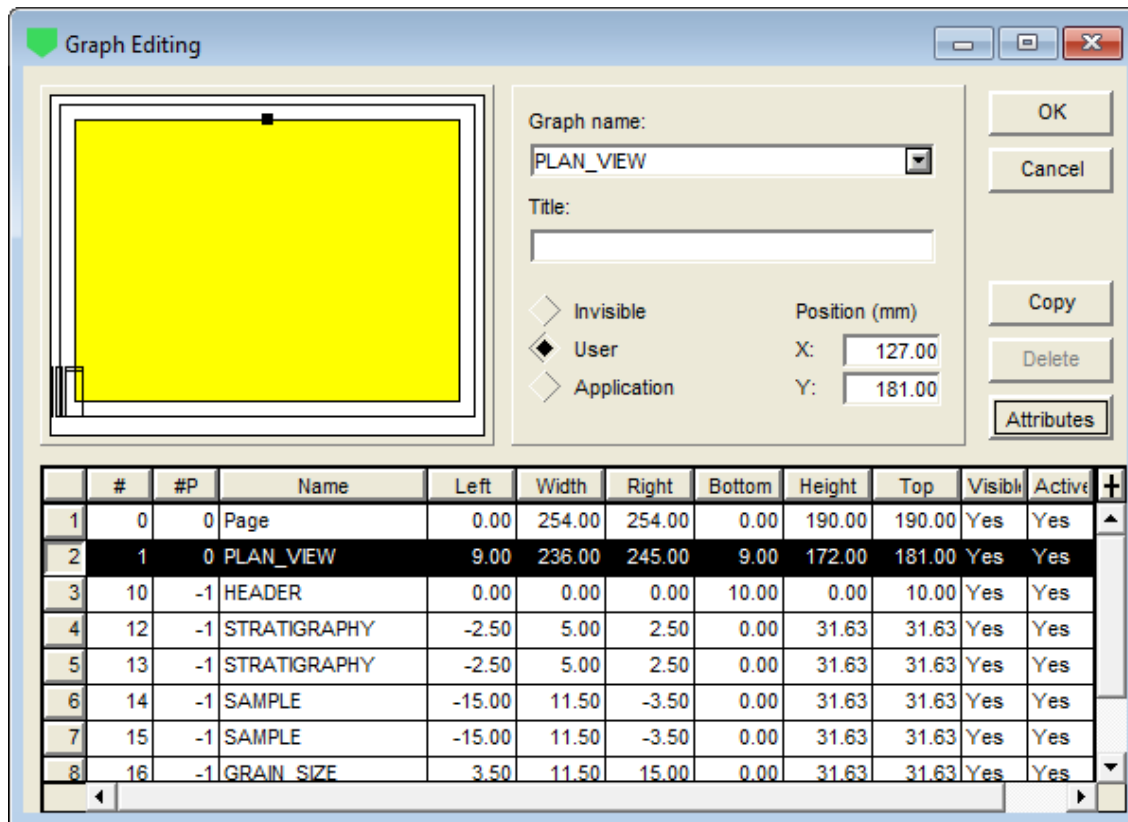


Figure 1-2 - Graph editing window

The **sub-graphs** are only displayed in the 3D view:

- The **HEADER** sub-graph displays the symbols and names of the borings.

The **Left**, **Width**, **Right**, **Bottom**, **Height** and **Top** fields reflect the position and size of the eventual identification boxes of the borings in the 3D view. See paragraph 9.4 for the details.

- The other sub-graphs show properties measured along the borings; for example, the stratigraphy, results of laboratory tests, results of in-situ tests such as the absorption etc.

These sub-graphs are considered property graphs; they are displayed as columns drawn parallel to the axis of the borings. The number and type of the property graphs are decided by the user; some of these graphs have predefined displays. The content of these graphs is defined in the legend and curves editing window. See paragraph 9 for details.

Each boring displayed in the 3D view will show the defined sub-graphs.

## 7. OVERVIEW OF GRAPHS

Site associates to certain graphs a display that is completely or partially predefined. These graphs are described in the table. The predefined curves have numbers greater or equal to **100**. Site manages the display of these curves, but the user can still modify some of the drawing attributes of the curves. Paragraph 4 of chapter 6 of the X3D guide explains how to modify the display attributes of a curve.

Graph	Characteristics
<b>PLAN_VIEW</b>	<p>The <b>PLAN_VIEW</b> primary graph shows the borings and axes in a plan view. It has ten (10) curves automatically generated:</p> <ul style="list-style-type: none"> <li>• The <b>Axis_Points</b> curve shows the turning points of the axes</li> <li>• The <b>Axis_Curve</b> curve shows the curve of the axes, their identification and their symbol</li> <li>• The <b>Axis_Profile</b> curve shows the position of the geophones and the profiles obtained</li> <li>• The <b>Boring</b> curve shows the numbers and symbols of the borings and their projection</li> <li>• The <b>Limit</b> curve shows the rectangular or polygonal limits</li> <li>• The <b>Dom_site_db</b> curve delimits the domain of the site as defined in the database</li> <li>• The <b>Dom_region_db</b> curve delimits the domain of the region as defined in the database</li> <li>• The <b>Dom_site_map</b> curve delimits the site map in the .sit file</li> <li>• The <b>Dom_region_map</b> curve delimits the region map in the .sit file</li> <li>• The <b>Dom_calculated</b> curve delimits the domain calculated according to the option selected in the options window</li> </ul> <p>Curves showing fields from the BORING table can be added to display information textually under each boring point</p>
<b>HEADER</b>	<p>The <b>HEADER</b> sub-graph, with its <b>Boring</b> curve, shows the symbols of the borings and their numbers.</p> <p>The <b>BORING</b> table provides information about the borings retrieved, including their number, their type (which influences the display of their marker or box) and their position.</p>
<b>STRATIGRAPHY</b>	<p>Typically, the <b>STRATIGRAPHY</b> sub-graph shows the symbols of the stratigraphic horizons (CODE1 field) or the color of their material (MATERIAL field).</p> <p>The STRATIGRAPHY sub-graph has predefined curves if the field displayed is CODE1, DEPTH_TOP or DESCRIPTION.</p>
<b>SAMPLE</b>	<p>Often, a <b>SAMPLE</b> sub-graph is defined to show the RQD of the cores.</p> <p>Special displays exist for the numbers and the state symbols of the samples.</p>
<b>BORING</b>	<p>The <b>BORING</b> sub-graph is mainly used to display the water levels, defined by <b>WATER_DEPTH</b> and <b>STABLE_WATER_DEPTH</b> and their respective dates.</p>
<b>PIEZOMETER</b>	<p>The <b>PIEZOMETER</b> sub-graph shows the installations of the piezometers, of the observation and supply wells and the measurements of the last piezometric levels.</p> <p>This sub-graph displays the contents of the <b>PIEZOMETER</b>, <b>PIEZOMETER_SEAL</b> and <b>PIEZOMETRIC_LEVEL</b> tables. It only contains predefined curves.</p>

Graph	Characteristics
<b>LIMITS</b>	The <b>LIMITS</b> sub-graph shows the measurements of the Atterberg limits. A predefined curve displays the plastic and liquid limits using their typical representation (vertical dashes joined by a horizontal line) and the water content represented by a circle.
<b>CONSOLIDATION</b>	The <b>CONSOLIDATION</b> sub-graph shows the results of consolidation tests. A predefined curve displays the minimum and maximum preconsolidation stresses using their typical representation (vertical dashes joined by a horizontal line) and the probable preconsolidation stress represented by a down arrow.
<b>ROCK_JOINT</b>	A special display exists to represent jointly the type, angle and azimuth of the joints, or to represent the angle of the joints.
<b>CPTU_ZONE</b>	The <b>CPTU_ZONE</b> sub-graph shows the ranges of SBT zones if the #1 curve is SBT. 9 predefined curves in the Page allow the user to define the colors of the 9 different ranges.

## 8. THE PRIMARY GRAPH

**Site** only has one primary graph: **PLAN\_VIEW**. It is described below. If the graph is set in 3D, it acts as a 3D graph. It is described in paragraph 8.2.

### 8.1. The **PLAN\_VIEW** graph

The **PLAN\_VIEW** graph shows the plan view of the selected site, region or domain. It should always be numbered #1. The plan view displays:

- the symbols of the borings with their identification as well as their projection, if applicable. The borings are displayed on the plan view if their coordinates are defined or if they are associated to an axis retrieved and have a chainage and an offset.
- information from the BORING table as texts, displayed under each boring point
- the axes retrieved with their identification next to their end points; an axe must have at least 2 turning points.
- a topographic background, if requested by the user. See paragraph 4.1.1.

#### 8.1.1. Predefined curves

Figure 1-3 shows the predefined curves (#100 to #134) of the **PLAN\_VIEW** graph. The display of these curves is not affected by their type.

- For each axis, the **Axis\_Points** curve shows:
  - markers at the turning points of the axis; these markers are different from the end points of the axis;
  - a line passing through the turning points of the axis; this line merges with the outline of the axis except in the curved sections of the axis;
  - the attributes that can be modified for the **Axis\_Points** curve are the line and the marker.
- For each axis, the **Axis\_Curve** curve shows:
  - its outline; this line follows the eventual curves defined by a radius at turning points;

- its identification; the name and symbol of the axis are displayed at both ends of the axis; the orientation of the text shows the origin of the axis: in reading position, the origin is left;
 

the marker used for each axis is the one in the current markers file whose name is the concatenation of the type and state of the axis, for example, LS\_R for a realised seismic line. If no corresponding marker is found, the one defined for the Axis\_Curve curve will be used. The color and dimension of the markers can always be modified.
- the attributes that can be modified for the **Axis\_Curve** curve are the line, the text and the marker (color and dimension, the symbol has no effect if there exists a marker for the type and state of the axis in the markers file).
- For each axis, the **Axis\_Profile** curve shows:
  - the location of the geophones; their location is shown by a marker;
  - the profile obtained; the thickness and color of the line can be modified by the user;
  - the attributes that can be modified for the **Axis\_Profile** curve are the line and marker.
- For each boring, the **Boring** curve shows:
  - its identification;
  - its symbol; the marker used for each boring is the one in the current markers file whose name is the concatenation of the type and state of the boring, for example, TP\_R for a realized test pit. If no corresponding marker is found, the one defined for the Boring curve will be used. The color and dimension of the markers can always be modified.
  - the projection of the boring on the axis, if the boring is inclined (with an azimuth and a dip);
 

this projection is a line starting at the center of the boring's symbol, oriented according to its azimuth, and whose length is the projection value of the boring onto the axis; a marker, as a dash, is displayed at the end of the projection line. If the user does not want to see the marker for the projection, he will set the period of the Boring curve to anything other than 1.
  - the attributes that can be modified for the **Boring** curve are the line (for the projection and its dash), the text and the marker for the boring (color and dimension, the symbol has no effect if there exists a marker for the type and state of the boring in the markers file).
- The **Limit** curve shows the rectangular or polygonal limits. The attributes that can be modified for the **Limit** curve are the line and the marker (for each corner of the limit).
- The **Dom\_site\_db** curve delimits the domain of the site as defined in the database. The attributes that can be modified for the **Dom\_site\_db** curve are the line and the marker (for each corner of the domain).
- The **Dom\_region\_db** curve delimits the domain of the region as defined in the database. The attributes that can be modified for the **Dom\_region\_db** curve are the line and the marker (for each corner of the domain).
- The **Dom\_site\_map** curve delimits the site map in the .sit file. The attributes that can be modified for the **Dom\_site\_map** curve are the line and the marker (for each corner of the domain).



- The **Dom\_region\_map** curve delimits the region map in the .sit file. The attributes that can be modified for the **Dom\_region\_map** curve are the line and the marker (for each corner of the domain).
- The **Dom\_calculated** curve delimits the domain calculated according to the option selected in the options window (see paragraph 4.1.1). The attributes that can be modified for the **Dom\_calculated** curve are the line and the marker (for each corner of the domain).

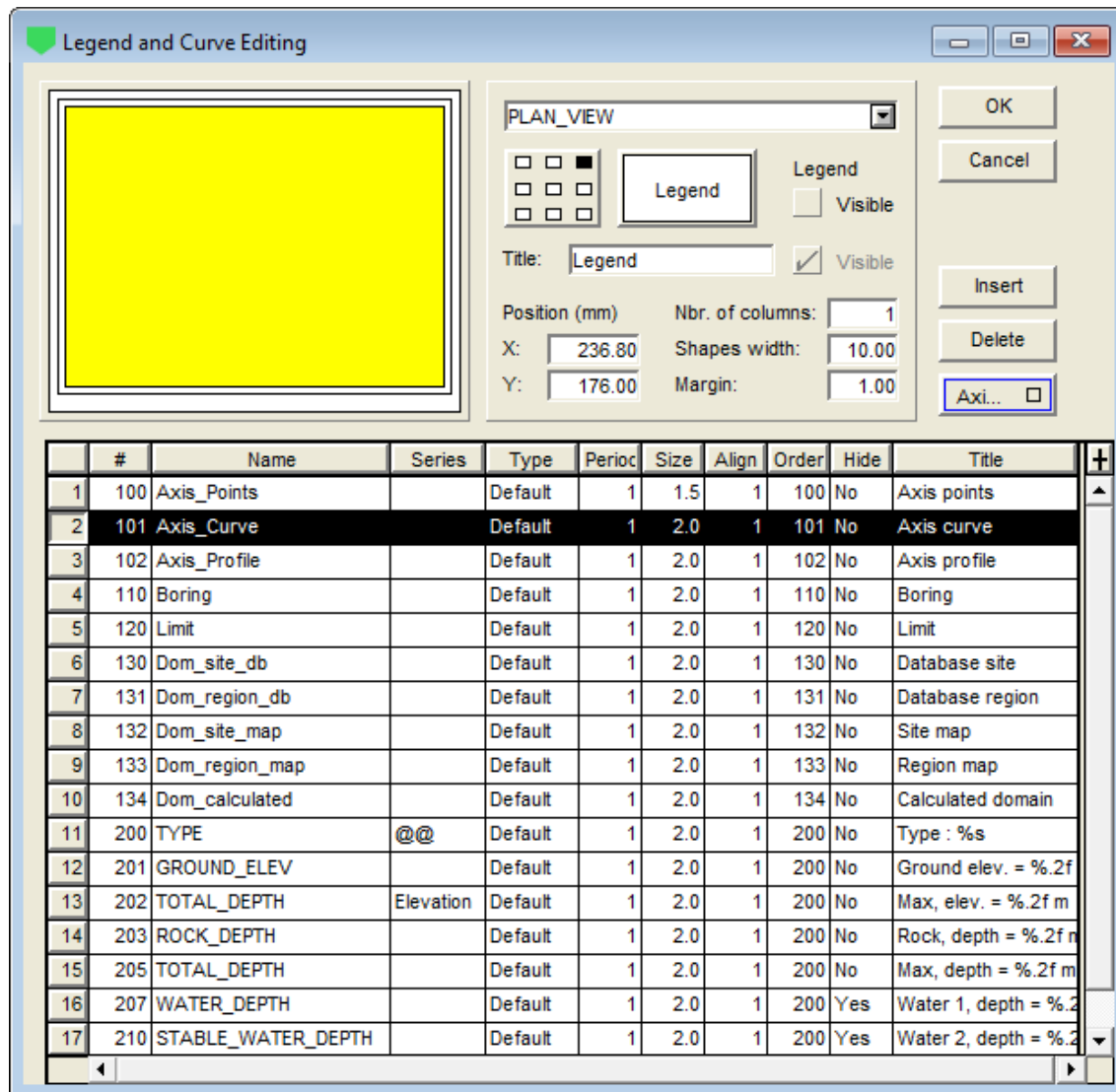


Figure 1-3 - Predefined curves of the PLAN\_VIEW graph

### 8.1.2. Curves of information

It is possible to display information textually under each boring present in the plan view, as illustrated in figure 1-4.

In the legend and curves editing window, for the PLAN\_VIEW graph, the user inserts a curve whose number # is 200 or more (see figure 1-3). In the scrolling list associated to the **Name** cell, the user chooses the field name of the BORING table to display. Only information from the BORING table can be displayed.



In the **Series** column, the user can select “Elevation” - the elevation will be calculated for the depth selected, or “@@” - the equivalence to the value in the EQUIVALENCE table or in the LIST\_ENG table will be displayed, if defined.

The user can modify the display format in the **Title** column as well as the drawing attributes for each curve. The information is displayed in ascending order of the curve numbers (#).


TP-12  
  
Type : Test pit  
Ground elev. = 410.20 m  
Max, elev. = 405.50 m  
Rock, depth = 4.70 m  
Max, depth = 4.70 m

Figure 1-4 - Example of textual information added in PLAN\_VIEW

## 8.2. The 3D\_VIEW

When the PLAN\_VIEW is set in 3D, it shows a tridimensional view. The 3D graph has the same curves as the PLAN\_VIEW graph, except the **Boring** curve. The domains are displayed at their smallest elevation. Note that if no elevation is found for the domain, it is displayed at Z = 0.

The 3D\_VIEW shows:

- the borings according to their dip and their sub-graphs of properties as columns parallel to the boring; the display of the properties of the borings is managed by the sub-graphs.
- the identifiers of the borings with their number and/or a marker characteristic of the type and state of the boring, or their number in a box.

if the markers are used, each boring will have as marker the one in the markers file whose name is the concatenation of the type and state of the boring, for example TP\_R for a realized test pit. If no corresponding marker is found, the one defined for the Boring curve of the HEADER sub-graph will be used (see paragraph 9.4).

for boxes to be used, their dimensions in millimeters must be defined by the user for the HEADER sub-graph (see paragraph 9.4); the test pits and trenches have rectangular or square boxes; the other borings have elliptical or circular boxes.

- the projections on a horizontal plan of the axes retrieved with their identification next to their ends; an axis must have at least 2 turning points.
- the profiles of the stratigraphic horizons measured using geophysical methods;  
the profiles are defined in the **AXIS\_PROFILE** table, each profile being identified by a type of material defined in the **AXIS\_STRATA** table. The color of each profile is the one associated to the material of code **S** defined in the **MATERIAL** table for the **SPEC** site. If the material is not defined in the **MATERIAL** table with the **S** code, the color of the **Axis\_Profile** curve is then used.
- a 3D topographic background, if requested by the user. See paragraph 4.1.1.

## 9. THE SUB-GRAPHS

The sub-graphs define the identification of the borings (HEADER sub-graph) as well as the columns of properties of the borings in the tridimensional view.

Figure 1-2 shows the graph and sub-graph editing window. Contrary to PLAN\_VIEW, the position and dimensions of the sub-graphs are not relative to the page.

The position of the top of each boring is its position in X, Y and Z, in the **3D view**.

The display of each boring represented in the 3D view is done according to its apparent dip, i.e. the tip of the boring's projection onto the plane perpendicular to the direction of observation of the 3D block. The angle and the length of the projection of each boring determine the angle and the length of the columns, therefore of the property graphs.

For the property graphs, the **Left**, **Width** and **Right** fields define the layout of the columns with respect to the axes of the borings and the width of these columns. A property graph with a value of 0.00 in the **Left** cell will lie against the right side of the boring. If a width of 0.00 is defined for a sub-graph, the column is not displayed. The **Bottom**, **Height** and **Top** fields are not used.

### 9.1. Creation of a property graph

A property graph is created in the graph editing window. More details about graph editing are available in paragraph 3 of chapter 6 of the X3D guide. Figure 1-2 shows the graph editing window in Site. To create a property graph:



- In the graph editing window:
  - Click on the **Copy** button, which creates a complete copy (with axes and curves) of the sub-graph selected, or substitute a graph by another.
  - Enter a **negative value** in the **#P** column of the graph.
  - In the scrolling list of the **Name** cell, select the name of the table from the Geotec database that contains the property to display. Many graphs can have the same name because they will not have the same number.
  - Define the position of the property column with respect to the borings in the **Left**, **Width** and **Right** cells.
  - Define the graph as **Visible**.
  - Click OK.



- In the legend and curves editing window:
  - Select the new property graph in the scrolling list at the top of the window.
  - Delete all the curves from the previous graph.
  - Click on **Insert** and choose the name of the field from the table whose data are to be displayed, in the scrolling list of the **Name** cell.
  - Choose the type of curve and define its drawing attributes by clicking on the button under Delete.
  - Click **OK**.
- Execute a new data query to fill the new graphs.

- In the axes editing window, configure the limits of the X-axis of the property graph.

## 9.2. Types of curves in the sub-graphs

There are several display types for the values in the property graphs: **Default**, **Text**, **Point**, **Histo**, **Step**, **Polyline**, **Polygon**, **Color** and **Pattern**. All types are described in details in paragraph 9 of the Log guide.

## 9.3. Format of the numerical values

The user can define the desired format for the display of numerical values in the **Format** field of the drawing attributes window for each curve; for example, "9.999" to have 3 decimals.

No format is predefined for texts placed as graphic objects representing numerical values from the database. The user can define a format right after the value string in the object's property window. For example, to display with 1 decimal number the dimensions of the site at the top of the report, the user will define a graphic object of **Text** type with the value **"&&SITE.WIDTH"9.9" , &&SITE.HEIGHT"9.9" "**.

## 9.4. HEADER sub-graph

The **HEADER** sub-graph is the graph for the identification and positioning of the borings in the tridimensional view. For each boring, the **HEADER** sub-graph can show:

- Its symbol (a marker or an identification box)
- Its number

The anchor point of the symbol of each boring is its position in X, Y and Z, in the **3D\_VIEW**.

To define the identifiers in boxes, the user will modify the **HEADER** sub-graph. The test pits and trenches have rectangular or square boxes; the other borings, elliptical or circular boxes. The **Left**, **Width**, **Right**, **Bottom**, **Height** and **Top** fields reflect the position and size of the boxes that include the identifiers of the borings:

- the **width** and the **height** define the size of the boxes, in millimeters; the two values must be positive.
- the **right** and the **left** indicate the position, in millimeters, of the right and left sides of the boxes with respect to the verticals at the implementation points of the borings.
- the **bottom** indicates at which distance, in millimeters, the bottom of the boxes will be located from the anchor points of the borings.

To display a marker instead of the boxes, the height or width of the **HEADER** sub-graph must be 0. The **bottom** indicates at which distance will be located, in millimeters, the center of the markers from the anchor points of the borings. The other values are not used.

For each boring, the **Boring** curve shows:

- the marker for the boring, if it has a positive size and if the **HEADER** sub-graph has at least one dimension at 0.

The marker is characteristic of the type and state of the boring; the marker used for each boring is the one in the current markers file whose name is the concatenation of the type

and state of the boring, for example TP\_R for a realized test pit. If no corresponding marker is found, the one defined for the **Boring** curve of the HEADER sub-graph is used.

The markers are centered on the verticals at the implementation points of the borings. Vertically, the centers of the markers are located at a distance indicated by the **bottom** value defined for the HEADER sub-graph, from the anchor points of the borings.

- the line between the anchor point and the **bottom** value of HEADER; if the boring number is in a box, the border of the box will have the same drawing attributes as the line.
- the boring number, with the text attributes.
- the button identified **Boring** opens the drawing attributes window, where the user can define how the boring numbers, the boxes and / or the markers (color) will be drawn. The marker defined for this curve will only be used for the borings for which the concatenation of the type and state does not correspond to any marker in the **.mrk** file used.

The drawing attributes of the **HEADER** sub-graph are not used.

## 9.5. STRATIGRAPHY sub-graphs

The stratigraphy of a boring is read in the **STRATIGRAPHY** table. Generally, the user displays the symbols representing the materials of the stratigraphic layers, by defining the CODE1 curve. Figure 1-5 shows the predefined curves associated to the CODE1 field. The user also often displays the color of the materials constituting the layers (MATERIAL field).

The content of the other fields of the STRATIGRAPHY table can also be displayed. The STRATIGRAPHY sub-graph has predefined curves if the field displayed is CODE1, DEPTH\_TOP or DESCRIPTION.

### 9.5.1. Layers and sub-layers

The layers and sub-layers are strata, which are records in the **STRATIGRAPHY** table.

- the strata can be classified as three **types**: **R** (rock classification), **S** (simplified classification) or **U** (unified classification);
  - for an **R** type stratum, only one material can be defined; its code is entered in **code 1**;
  - for an **S** type stratum, up to six materials can be defined; their codes (one character each) are entered in **code 1**;

the relative proportions of the materials are entered in the **proportion** field; the main material has a proportion of 1, a material with traces has a proportion of 4; the proportions must be entered in the same order as the materials in code 1; more than one material can have the same proportion;

**code 2** is used to indicate grain size or plasticity characteristics, it does not have an impact on the display;

the **line** can be used to force the display of the top horizon limit using a specific format; the type of line requested will prevail over the one used for the lines belonging to the curve; the types of line that can be imposed are:

- **L0** blank line;
- **L1** continuous line;
- **L2** dotted line;
- **L3** dashed line;

the values of L0 to L3 entered in **code3** in the previous versions of Geotec will be used if there are no values in Line.

- for a **U** type horizon, two materials can be defined in **code 1** from the scrolling list; they are separated by a dash;
- each layer is limited by top and bottom depths and can have a text description.

In addition to the strata:

- descriptions can be added at specific depths along the boring; these records of type **T** (text) have a top depth but no bottom depth;

if a record of type **T** has a negative depth, its description is displayed above the top line of the column, generally the ground surface;

if there is no record of type **T** at a negative depth, Site automatically displays, at the top of the log, the title of the **Elevation** curve of the **STRATIGRAPHY** sub-graph associated to the **DESCRIPTION** field;

- a description can be associated to the bottom of the boring; this record of type **E** has a top depth but no bottom depth;

if there is no record of type **E**, Site automatically displays at the bottom of the column the title of the curve **End\_of\_borehole** of the **STRATIGRAPHY** sub-graph associated to the **DESCRIPTION** field;

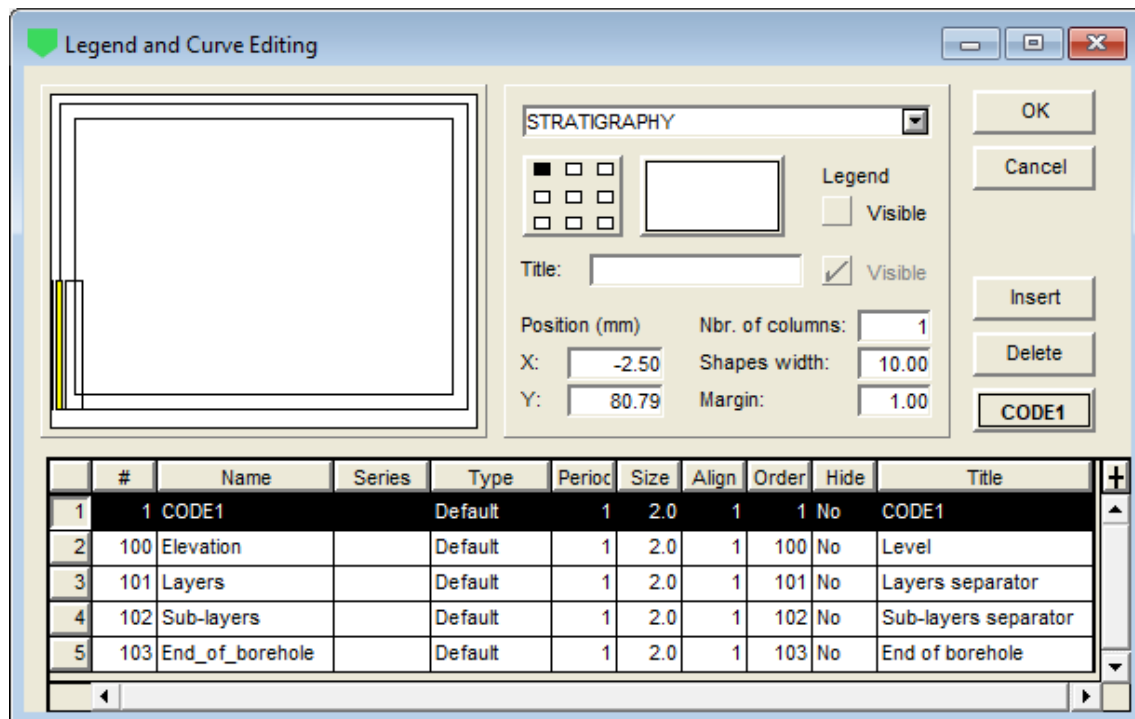


Figure 1-5 - Predefined curves of a STRATIGRAPHY sub-graph

A stratum is a **layer** if its classification type, its code 1 **or** its proportions (for the S type) is different than the preceding stratum.

A horizon is a **sub-layer** if its classification type, its code 1 **and** its proportions are the same as those from the preceding stratum.

The **Layers** and **Sub-layers** curves affect the stratigraphic layers and sub-layers.

### 9.5.2. Display of the STRATIGRAPHY sub-graph for the CODE1 field

Depending on the field specified as the non-predefined curve of the **STRATIGRAPHY** sub-graph, the display associated to the curves varies and depends on Site.

The only case described in this guide is the one where the field entered as the non-predefined curve is **CODE1**, which is the most common case. The reader can find information in [paragraph 13 of the Log guide](#) about the display particularities when the field entered as the non-predefined curve is **DEPTH\_TOP**, **DESCRIPTION**, **MATERIAL** or another field.

If the **CODE1** curve is of **Default** type:

- The column shows the stratigraphic symbols in graphic format if they are defined in the current **.ptn** pattern file, otherwise using a text format (see paragraph 9.5.3 about the display of stratigraphic symbols).
- The **Elevation** curve displays nothing.
- The **Layers** curve displays the upper limit of the layers.
- The **Sub-layers** curve displays the upper limit of the sub-layers.
- The **End\_of\_borehole** curve displays the end of boring limit and, if the record of type **E** has a **code 1** of **R**, the bedrock symbol (illustrated below) indicating the refusal (or stop) of the boring on the bedrock, is drawn with the drawing attributes of the curve.



If the **CODE1** curve is a **Text**, the column displays the stratigraphic symbols using a text format and the predefined curves are not considered. The attributes of the limits of the layers and sub-layers and of the textual stratigraphic symbols are those defined for the **CODE1** curve.

If the **CODE1** curve is of a type other than **Default** or **Text**, nothing is displayed in the column. The types of the predefined curves are irrelevant.

### 9.5.3. Display of the stratigraphic symbols

To graphically display a stratigraphic symbol, an X3D pattern must be associated to the material code to be represented in the pattern file (**.ptn**) used (and the **CODE1** curve must be of **Default** type). Otherwise, the material will be represented textually.

- If a record is of type **R**, its code 1, for example **CAL**, will be represented graphically if there is a pattern named **R\_CAL**, according to our example.
- If a record is of type **S**, the graphic representations of the materials specified in code 1 will be superimposed if a pattern exists for each of these codes, whose name begins with **S\_** followed by the code to be represented and the proportion of the material;

for example, the **CMG** code with a proportion of **123** will be represented by three superimposed patterns if the patterns **S\_C1**, **S\_M2** and **S\_G3** exist in the **.ptn** pattern file being used.

- If a record is of type **U**, if a pattern exists for the complete **code1**, for example ML-MH, and whose name is **U\_ML-MH**, it will be used for the graphic representation.

The **geotec8.ptn** file provided with the installation contains patterns for most of the standard codes used. The user can refer to the X3D guide for information about how to look up, edit and create patterns.

## 9.6. SAMPLE sub-graphs

Often, a **SAMPLE** sub-graph is defined to show the **N** or **RQD** of the samples. The content of the other fields of the **SAMPLE** table can also be displayed. The curves **SAMPLE\_NO**, **STATE**, **USCS** and **MATERIAL** have a special display.

### 9.6.1. Display of the **SAMPLE\_NO** curve

If the **SAMPLE\_NO** curve is a **Text**, only the sample number is displayed.

If the **SAMPLE\_NO** curve is of **Default** type, it shows the sample numbers concatenated to their type, for example SS-01; this layout cannot be modified.

The user can modify the attributes of the text and line showing the top and bottom depths of each sample.

### 9.6.2. Display of the **STATE** curve

If the **STATE** curve is a **Text**, the states of the samples are represented by their textual code.

If the **STATE** curve is of **Default** type, a graphic symbol represents the state of the samples along their entire length.

Four symbols, shown in figure 1-6, are generally used to represent the state of the samples. To display these symbols, the drawing attributes of the **STATE** curve must be defined as follows:

- The **Line** is defined with the color, type and size desired; the line attributes apply to all states; the line is used as the symbol border and for the cross, for a **remolded** sample;
- The **Pattern** must contain an **X3D** pattern (usually a hatched pattern); this pattern applies only to **Intact** samples.
- The **Filling** applies to the **Lost** and **Core** samples; it must be defined in two steps:
  - It must be defined with the desired color;
  - It must be "erased" with the "-" button for the filling not to affect the intact samples.

The **remolded** samples are the only ones with a symbol set by the **Site** module; only the line attributes forming the cross can be modified.

The **lost percentage** of a sample can be graphically represented at the bottom of the sample with a height proportional to the lost percentage. To do this, indicate a **period of 2** for the



STATE curve. The filling attributes will be used. If a sample has a **recovery of 0%**, it is drawn as **'lost'**.

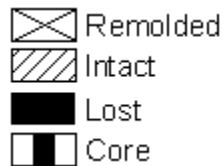


Figure 1-6 - Symbols representing the state of the samples

### 9.6.3. Display of the USCS curve

If the **USCS** curve is of **Default** type with a **period of 1 or less**, the column shows the X3D patterns defined in the current **.ptn** patterns file, named "**U\_USCS**", for example, **U\_GW**.

If the **USCS** curve is of **Default** type with a **period greater than 1**, the column shows the colors associated to the materials of code **U** defined in the **MATERIAL** table for the **SPEC** site.

If the **USCS** curve is a **Text**, the column shows the USCS classification in text format. The attributes of the text and of the limits between the samples are defined for the **USCS** curve.

If the **USCS** curve is of a type other than **Default** or **Text**, nothing is displayed in the column.

### 9.6.4. Display of the MATERIAL curve

If the **MATERIAL** curve is of **Default** type, the column shows the background colors for the materials constituting the samples (materials of the borrow pits). The colors are the ones associated to the materials of code **S** defined in the **MATERIAL** table for the **SPEC** site.

If the **MATERIAL** curve is a **Text**, the column shows the codes of the materials in text format. The attributes of the textual materials and of the limits between the layers are defined by the **MATERIAL** curve.

If the **MATERIAL** curve is of a type other than **Default** or **Text**, nothing is displayed in the column.

### 9.6.5. Selection of the types of samples to present

**Site** allows the user to restrict the display of the samples to just one type. To do this, indicate the type of samples to display in the **Series** column; for example, "**SS**" for split spoons.

Each cell of the **Series** column has a scrolling list which displays all the types of samples inserted in the current database.

The user can select a different type for each curve of a **SAMPLE** sub-graph. Only the values associated to the selected type will be displayed. If the cell of the **Series** column is empty or if it contains "**@@**", all the samples will be represented for this curve.

## 9.7. PIEZOMETER sub-graph

The installations of the piezometers, observation and supply wells as well as the seals, lanterns and piezometric levels are shown in the **PIEZOMETER** sub-graph.



Site reads the information about the piezometers in the **PIEZOMETER** table, the information about the seals and lanterns in the **PIEZOMETER\_SEAL** table and on the piezometric levels in the **PIEZOMETRIC\_LEVEL** table.

The **PIEZOMETER** sub-graph can only contain the predefined curves. Paragraph 15 of the **Log guide** gives a complete description of each predefined curve of **PIEZOMETER**.

## 9.8. LIMITS sub-graph

The **LIMITS** sub-graph displays the results of the Atterberg limits measurements: the plastic limit, the liquid limit and the water content. **Site** reads the data in the **LIMITS** table.

When a **LIMITS** sub-graph is created, the predefined curve **PL-LL** is created. This curve shows a typical display, i.e. two vertical dashes representing the plastic and liquid limits and joined by a horizontal line, as well as an empty circle representing the water content. The symbols are positioned at the depth corresponding to the center of the samples.

The user can open the drawing attributes window and modify the **marker**, which will then be the symbol for the water content. The **line** can be modified; it will have an effect on the horizontal line and the vertical dashes for PL and LL. The **text** is displayed by default, but can be modified or removed ("- button) via the attributes window.

If the user does not want to display any marker, he will enter a size of 0 for the PL-LL curve. If he only wants the markers for the limits, he will enter a period of 0 in the attributes window for the curve.

It is also possible to add curves for the **W**, **LL** and **PL** fields to impose a specific display; for example, to connect all the plastic limits, the liquid limits and the water contents, select the **Polyline** type for these curves. Each polyline is defined by the **line** attribute. If the **W**, **LL** and **PL** curves are of **Point** type, only the markers will be displayed.

## 9.9. CONSOLIDATION sub-graph

The **CONSOLIDATION** sub-graph displays the measurements of consolidation pressures read in the **CONSOLIDATION** table.

When a **CONSOLIDATION** sub-graph is created, the predefined curve **PCMIN-PCMAX** is created. This curve shows a typical display, i.e. two vertical dashes representing the minimum and maximum preconsolidation stresses and joined by a horizontal line, as well as an empty down arrow representing the probable preconsolidation stress. The symbols are positioned at the depth corresponding to the center of the samples.

The user can open the drawing attributes window and modify the **marker**, which will then be the symbol for the probable PC. The **line** can be modified; it will have an effect on the horizontal line and the vertical dashes for the minimum and maximum preconsolidation stresses. The **text** is displayed by default, but can be modified or removed ("- button) via the attributes window.

If the user does not want to display any marker, he will enter a size of 0 for the PCMIN-PCMAX curve. If he only wants the markers for the minimum and maximum stresses, he will enter a period of 0 in the attributes window for the curve.

It is also possible to add curves for the other fields of the table, the same way as for the curves of the LIMITS sub-graph (see paragraph 9.8).

### 9.10. ROCK\_JOINT sub-graphs

All the fields from the **ROCK\_JOINT** table can be displayed in curves of a **ROCK\_JOINT** sub-graph.

The **ALPHA** (or **ANGLE** in the previous versions of Geotec), **DIP** and **TYPE** curves have a special display. These three curves cannot be displayed in the same sub-graph.

The curve associated to the **ALPHA** (or **ANGLE**) field has a specific display if it is of **Default** type. It then displays the joints as lines inclined according to their angle relative to the core axis and centered on their average depth. The lines have a length of 80% of the column's width. The elements that can be edited for the **ALPHA** (or **ANGLE**) curve depend on the type of curve chosen.

The display is similar for the curve associated to the **DIP** field, except that the line is inclined relative to the horizontal, in counter-clockwise direction.

The curve associated to the **TYPE** field is used to display information relative to three fields: **TYPE** and of one of the following pairs depending on the reference taken for the measurement of the angles: **ALPHA** and **BETA**, or **DIP** and **DIRECTION**. If a marker named **J\_type** is defined and an angle is entered in the database for the same depth, the marker representing the type will be displayed at the location of the value between 0 and 90 on the X axis, and will be inclined according to the azimuth value (between 0 and 360). If no azimuth value is defined, a value of 0 (vertical) will be used. For this display, the **TYPE** curve must be of **Default** type. If the curve is of **Text** type, only the type values will be displayed textually.

### 9.11. CPTU\_ZONE sub-graphs

It is possible to display the ranges of SBT zones calculated as histograms or as textual description for the results of piezocone tests. For this, the curve #1 of the CPTU\_ZONE sub-graph must be associated to the **SBT** field.

- If the **Series** cell of the SBT curve is empty, the column will display the ranges of SBT zones as histograms. The period for the curve must be 0 or a filling must be defined for the curve to show the different colors for the ranges.
- If the **Series** cell of the SBT curve contains "@@", the column will display the SBT ranges as descriptions. If the **EQUIVALENCE** table contains the description of the zones as equivalences to their value, this description is displayed; otherwise, the numbers of the zones are displayed.

Nine predefined curves are automatically included in the **Page** graph when the SBT curve of CPTU\_ZONE is displayed. These curves named "**Zone\_1**" to "**Zone\_9**" are used to define the colors of the various values of SBT zones. A series of colors commonly used is proposed by default. However, the user can modify these colors by editing the "**Zone\_n**" curves.

### 9.12. Display of property values or of concentrations of contaminants

The **PROPERTY\_VALUE** table contains the values of various properties according to depth; the properties are defined by the user. Also, the **CONCENTRATION** table contains the

concentrations according to depth of various contaminants defined in the **CONTAMINANT** table.

To display the values of the **PROPERTY\_VALUE** table for a specific property in a column, the user must:

- associate the **PROPERTY\_VALUE** table to a sub-graph;
- define at least one curve associated to a field of the table in the curve editing window; usually, it is usually the **CVALUE** field.
- in the **Series** column of the curve definition, enter the property for which values are to be represented;

The display in a column of values from the **CONCENTRATION** table for a contaminant is done the same way, with the name of the **contaminant** in the **Series** cell.

### 9.13. Display of elevations and information associated to a depth curve

In a sub-graph other than the ones described above, we can define curves associated to a field starting with **DEPTH\_** or finishing with **\_DEPTH**. These curves must be of **Default** type. We can add information at each depth in the sub-graph by entering the information to be displayed in the title of the **DEPTH\_xxx** or **xxx\_DEPTH** curve.

The information displayed can be any text. It can contain the value of a field from a **Geotec** table; the syntax is then **&&table\_name.field\_name"format"** to display the values, or **@&&table\_name.field\_name"format"** to display the values or eventual equivalences (see paragraph 9.14). The format is not required.

For example, the user can create a column to display the initial and stabilized depths of the water table using a marker and the reading date. For this:

- the user associates the **BORING** table to a sub-graph;
- the user defines two curves. The first curve is associated to the **WATER\_DEPTH** field and has this expression in the **Title** cell: **&&BORING.WATER\_DATE"yyyy-mm-dd"**. The second curve is associated to the **STABLE\_WATER\_DEPTH** field and has this expression in the **Title** cell: **&&BORING.STABLE\_WATER\_DATE"yyyy-mm-dd"**. A marker for the water level can be associated to each curve.

It is also possible to display the elevations corresponding to the depths presented. These elevations are displayed if the **ELEVATION** keyword is specified in the title of the curve. These keywords cannot be included in a string. The dip of the boring is taken in account to determine the value of the elevations.

If no title is provided for a **DEPTH\_xxx** or **xxx\_DEPTH** curve of **Default** type, the depth is then displayed.

If the user wants to display the depths and the elevations, two **DEPTH\_xxx** or **xxx\_DEPTH** curves of **Default** type must be created in the sub-graph. One of the titles will remain empty, while the other will be **ELEVATION**. By aligning the characters of a curve by the top and the other by the bottom, both texts will not be superimposed in the graph.

### 9.14. Display of equivalences to values

It is possible to display equivalent expressions to some values in a curve if the following conditions are met:

- The curve does not belong to a sub-graph with **predefined curves**;
- The curve does not belong to a sub-graph representing the **CONCENTRATION** or **PROPERTY\_VALUE** table.
- The curve does not represent a **DEPTH\_xxx** or **xxx\_DEPTH** field;
- Expressions equivalent to one or many values are defined in the **EQUIVALENCE** table or in the **LIST\_ENG** table for the field represented by the curve;
- The @@ expression is entered in the **Series** column of the curve.

Regardless of whether the field values are numerical or textual, if the equivalent expressions are textual, they will only be displayed if the curve is of **Default** or **Text** type. If the equivalent expressions are numerical, they can be displayed regardless of the type of curve.

## 10. INTERACTIVE EDITING

### 10.1. Interactive editing in 3D view

In **3D mode**, the user can click with the right button of the mouse to display the contextual menu shown in figure 1-7. **Create** and **Information** are active only if the mouse is over a property column.

The first item is used to **create** a new record in the table whose data are displayed in the column. The depth clicked will be displayed by default for the new record.

The **Information** option opens the correct entry form on the data of the record where the user clicked. Double-clicking on data has the same role.

With the interactive menu, it is also possible to launch the modules of Geotec to show the various types of data that can be displayed for the current boring, site or axis.

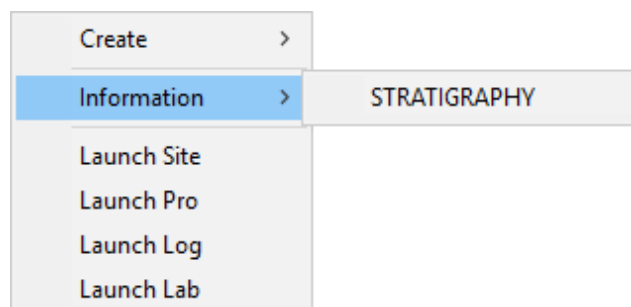


Figure 1-7 - Interactive menu in 3D view

### 10.2. Interactive editing in 2D view

In **2D mode**, the user can right click anywhere in the plan view outside the geotechnical entities to display the contextual menu shown in figure 1-8.

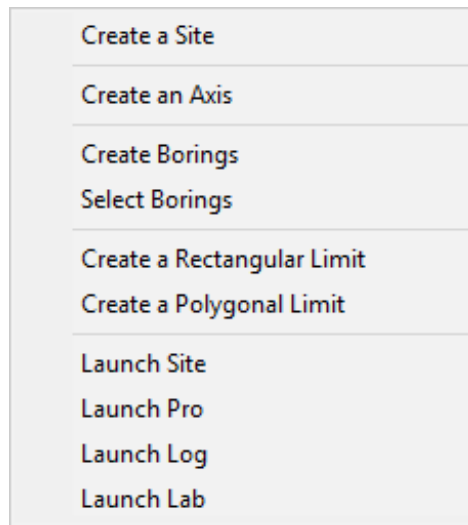



Figure 1-8 - Interactive menu in 2D view


### 10.2.1. Creating a site

By selecting **Create Site**, the cursor takes this format: . If the user wants to cancel right away, right click and select **Cancel**. To create the site, left click at one corner of the new site, hold down and move the cursor to the diagonally opposite corner; a dotted line shows the area created. When the button of the mouse is released, the site entry form is displayed.


By default, the MTM and NAD83 systems are used. The X and Y coordinates of the bottom left corner, its longitude and latitude, the zone used for the calculation, as well as the width and height of the area are automatically entered. If Google Map is displayed in background, the zone is determined from there. Otherwise, the zone used is the site's zone previously displayed.


The user then enters the site number and any other information in the entry form, and saves the new site.

### 10.2.2. Creating an axis

By selecting **Create Axis**, the cursor takes this format: . Left click at each point of the axis. A dotted line shows the movement of the cursor. When the last point of the axis is defined, right click to display the pop-up menu and select **Cancel** to erase the axis being created, or **Terminate** to edit and save the new axis. The axis entry form is then displayed and shows the coordinates of each point clicked. The user can then enter the axis number and any relevant information in the entry form, and save.

### 10.2.3. Modifying an axis

When the cursor is on an axis, it takes this format: . By right clicking, the menu shown in figure 1-9 is displayed.

The first 2 items are inactive except for **planned** axes (state of axis: P). With **add an axis point**, the cursor becomes . The point will be added between the two closest points to the cursor. A message is displayed, showing the exact coordinates of the point. To stop the addition of points, right click and select **Terminate**. When the cursor is over an axis point and there are

more than two points in the axis, the option **Delete this Axis Point** is active. A message is displayed to confirm the deletion of the point. When the cursor is on an axis point, it takes this format: +<sup>+</sup>. By clicking on the point, it takes the format <sup>+</sup>. Move the point to the desired position, and release the button of the mouse; a message is displayed to confirm the move.

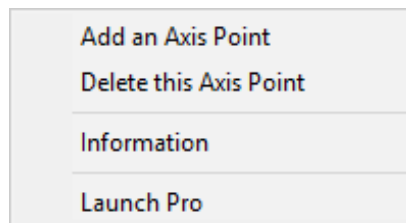
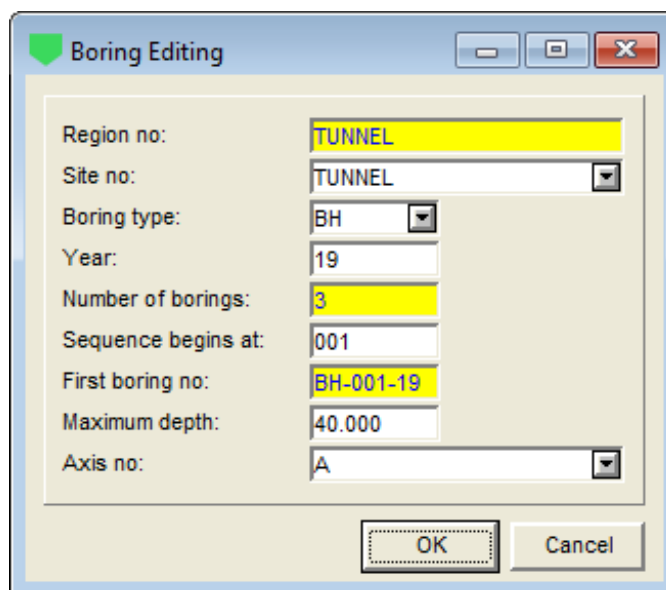


Figure 1-9 - Editing an axis

The **Information** option opens the axis entry form on the current axis. It is also possible to launch the **Pro module** to show the profile of the current axis.

#### 10.2.4. Creating borings

By selecting **Create Borings**, the cursor takes this format: +<sup>+</sup>. Left click at the location of the first boring. A dotted line shows the movement of the cursor. Left click as many times as desired to add borings. When the last boring is defined, right click to display the pop-up menu and select **Cancel** to erase all borings being created, or **Terminate** to edit and save the new borings. The window shown in figure 1-10 is then displayed.



The Boring Editing dialog box contains the following fields and controls:

- Region no: TUNNEL
- Site no: TUNNEL
- Boring type: BH
- Year: 19
- Number of borings: 3
- Sequence begins at: 001
- First boring no: BH-001-19
- Maximum depth: 40.000
- Axis no: A
- Buttons: OK, Cancel

Figure 1-10 – Creating new borings

The boring editing window shows:



- The **region number**, associated to the site selected.
- The **site number**. The user can select another from the list of all sites currently displayed.
- The **boring type**. The user can select another from the list of types of borings that have a defined maximum depth.

- The **year** of the projected borings, as “yy”. This is added to the boring numbers.
- The number of borings clicked.
- The number for the **beginning of the sequence**. By default, the number is one greater than the largest boring number found in the site selected.
- The **first boring number**. The number is a concatenation of the type - sequence - year.
- The **maximum depth** of all planned borings.
- The **axis number** to which the borings are attached.

When the user clicks OK, the borings are added in the plan view using the marker for the corresponding type of planned borings, and their coordinates are also entered in their record.

### 10.2.5. Modifying a boring

When the cursor is on a boring, the contextual menu shown in Figure 1-11 is displayed when right clicking.

The first item is inactive except for **planned** borings (state of boring: P). When the cursor is over a planned boring, the option **Delete this Boring** is active. A message is displayed to confirm the deletion. When the cursor is on a planned boring, it takes this format: +. By clicking on the boring, it takes the format +. Move the boring to the desired position, and release the button of the mouse; a message is displayed to confirm the move.

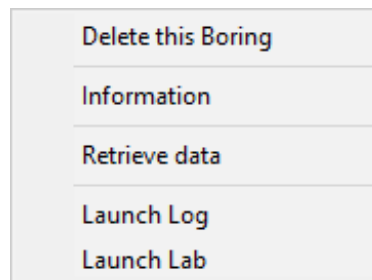



Figure 1-11 - Editing a boring

The **Information** option opens the boring entry form on the current boring. It is also possible to launch the **Log** or **Lab modules** to show data from the current boring.

### 10.2.6. Selecting borings

By clicking **Select Borings**, the cursor takes this format: . Click and drag a rectangle containing the desired borings. The borings selected are then displayed in blue. The user may also select or unselect borings by pressing the [Ctrl] key and clicking on the desired borings.

When all desired borings are selected, right click to display the menu shown in figure 1-12. The option **Create a Group of Borings** will open the selection entry form; each boring selected is identified with its site and boring number. A selection number corresponding to the current boring number is displayed by default, and can be modified. The user then enters any other information in the entry form, and saves the new group of borings.



With the option **Delete these Borings**, all borings selected and whose state is **Planned** will be deleted, after confirming the message displayed. Records for borings of another state cannot be deleted this way.

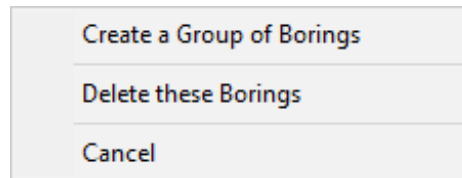





Figure 1-12 - Selection of borings


### 10.2.7. Creating and modifying a rectangular limit

By selecting **Create Rectangular Limit**, the cursor takes this format: . If the user wants to cancel right away, right click and select **Cancel**. Left click at one corner of the rectangular limit, hold down and move the cursor to the diagonally opposite corner; a dotted line shows the area created. When the button of the mouse is released, the limit entry form is displayed; the X and Y coordinates of the four corners of the limit are automatically entered. The current site number is displayed as reference site. The user enters the limit number and any other information, and saves the new rectangular limit.



### 10.2.8. Creating and modifying a polygonal limit

By selecting **Create Polygonal Limit**, the cursor takes this format: . If the user wants to cancel right away, right click and select **Cancel**. Left click at each point of the polygonal limit. A dotted line shows the movement of the cursor. When the last point of the polygonal limit is defined, right click to display the pop-up menu and select **Terminate**. The limit entry form is then displayed; the X and Y coordinates of all points of the limit are automatically entered. The current site number is displayed as reference site. The user enters the limit number and any other information, and saves the new polygonal limit.

Any rectangular or polygonal limit can be modified with the mouse. When the cursor is on a limit, it takes this format: . By right clicking, a contextual menu is displayed.

When **adding a point to the limit**, the cursor becomes . The point will be added between the two closest points to the cursor. A message is displayed, showing the exact coordinates of the point. To stop the addition of points, right click and select **Terminate**.

When the cursor is over a point of the limit, the option **Delete this Point from Limit** is active. A message is displayed to confirm the deletion of the point.

When the cursor is on a point of the limit, it takes this format: . By clicking on the point, it takes the format . Move the point to the desired position, and release the button of the mouse; a message is displayed to confirm the move.



## CHAPTER 2. SM2 - SITE MAPPING AND MODELING

The SM2 library is primarily used to edit topographic maps, to produce data terrain models (DTM) by dual kriging or triangulation, to perform mathematical operations on DTMs and to establish iso-values contours.

SM2 uses a Graphical User Interface (i.e. with windowing) that is compatible with Microsoft Windows 7, 8 and 10. SM2 uses the X3D graphic library created by Sobek Technologies Inc. to define the presentation style, view and edit graphic results. Please refer to the X3D guide for all details.

---

### 1. LAUNCHING THE SM2 LIBRARY



The **SM2** library is integrated in the Site module, opened by double-clicking on this icon. The **SM2 Mode** option of the Tools menu or this button opens the SM2 library. Site always opens in the last mode used (Site or SM2).

No database is necessary for with SM2. When **Site** or **SM2** is first launched, the following files are loaded from **C:\Geotec80\Style**:

- the markers file **geotec8.mrk**;
- the patterns file **geotec8.ptn**;
- the style file **site\_landscape.sty** which is a classic presentation model showing a plan view.

The user can begin working with this initial presentation, or open another style file included with the software. He can modify the style file and save it in a new presentation. In subsequent loadings, Site opens the style, markers and patterns files that were in use when Site was closed.

---

### 2. FILE, EDIT, INSERT, WINDOW AND HELP MENUS

None of the options of the **File**, **Edit**, **Insert**, **Window** and **Help** menus is specific to SM2. The detailed description of each option of these menus is done in chapters 2, 3, 4 and 8 of the X3D guide.

---

### 3. VIEW MENU



The **2D View** and **3D View** options of the **View** menu and this button are used to change the view of the PLAN\_VIEW graph. In the axes editing window, it is possible to change from 2D to 3D view and vice-versa for PLAN\_VIEW.



The **OpenGL mode** is used to view solids and surfaces with hidden lines and faces and shadows in 3D. The **XVT mode** is the standard display mode.

## 4. TOOLS MENU

The options of the **Tools** menu are used to define the utilisation parameters of SM2 and to edit certain graphic components produced by SM2: the characteristics of the graphic page, the PLAN\_VIEW graph, the axes, the markers and patterns used. All these options are described in details in chapter 6 of the X3D guide. **Options** is not used in SM2.



The **SM2 Mode** option is checked when the SM2 library of site mapping and modeling is being used. By unchecking the option or the button, SM2 is closed and switched to the Site module for the management of geotechnical data.

## 5. ORGANIZATION OF A GRAPHIC OUTPUT



The definition of graphs is established and updated in the graph editing window opened by selecting the **Graphs...** option of the **Tools** menu or by clicking this button. More details about the graph editing are available in paragraph 3 of chapter 6 of the X3D guide.

Each graph is identified by a unique number in the **#** column. The first graph is always **Page** with **# 0**. This graph represents the graphic page and cannot be deleted.

In SM2, only the **PLAN\_VIEW** graph is used, and shows the content of the data file opened or of the files produced by the various modeling operations. This graph can be set in 2D or 3D, in XVT or OpenGL mode. The **Left**, **Width**, **Right**, **Bottom**, **Height** and **Top** fields define its position and dimensions; the user can modify these values.

## 6. MAPPING MENU

The **Mapping** menu, shown in figure 2-1, is specific to the SM2 library. The menu options are used to edit the content of cartographic files. These do not include “grid” files such as the .dem, .dtm, .k2d, .tin, etc. files.

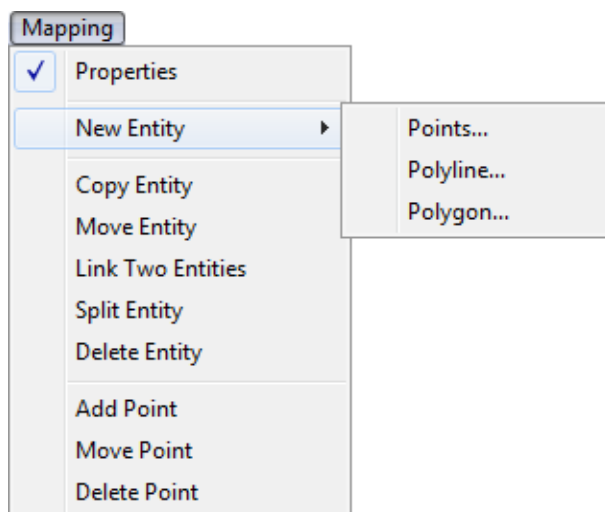



Figure 2-1 - Mapping menu

## 6.1. Properties and editing of a cartographic entity



The **Properties** option of the menu is checked by default, until another option is selected.

The user can left click on any entity in the cartographic file. The entity is then highlighted and the cursor takes the shape . The highlighting attributes can be modified (see paragraph 7.1.8). By holding down the left button of the mouse, some properties of the entity are displayed in the status bar: the name of the entity's layer, the number of the point left of the location clicked (left is towards the 1<sup>st</sup> point of the entity), the total number of points, and the proportion factor indicating the distance between the point to the left and the location clicked, between 0 and 1. Figure 2-2 shows an example.

By double-clicking on the entity, a property window is opened, as illustrated in figure 2-3. The **Properties** option of the popup menu displayed with the right button of the mouse is equivalent.

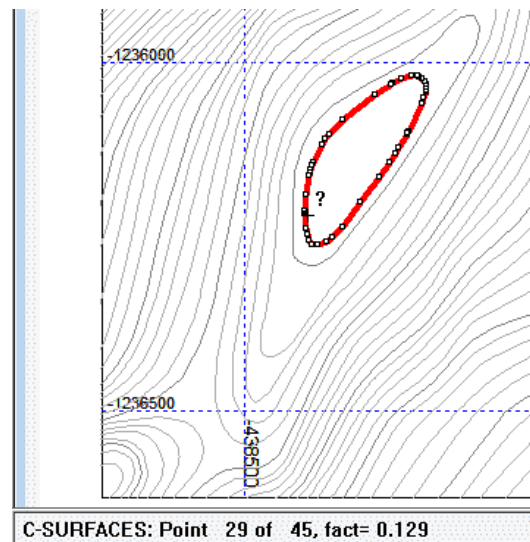


Figure 2-2 - Example of a highlighted entity and its properties

### 6.1.1. Name of layer

The **Layer** field contains the name of the layer to which the cartographic element belongs. This value can be modified. The scrolling list contains the names of all the layers belonging to the current data file.

If the user enters a new layer name, the entity will belong to it and this new layer will be added, with the attributes of the current element. The user will be able to edit it via the **Layer Editing** option of the **Modeling** menu and to save it by saving the data file. See paragraph 7.2 for the layer editing.

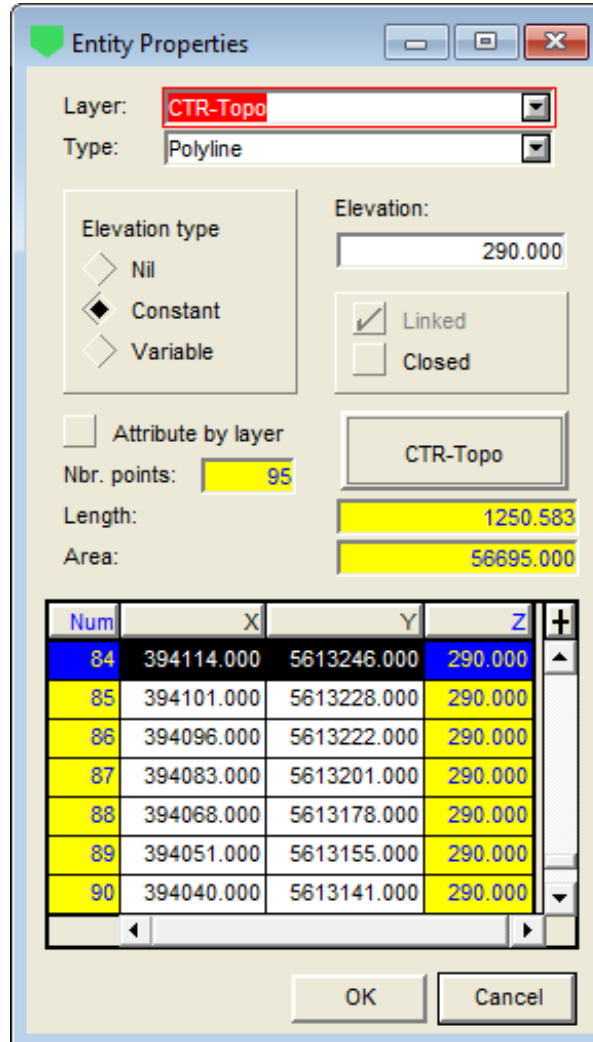
### 6.1.2. Type of entity

The **Type** field contains the type of the cartographic element. The scrolling list contains all the possible types of elements: **Polyline**, **Points** and **Polygon**.

### 6.1.3. Elevation of the points

The user can decide on the **Elevation type**.

- **Nil**: the elevation of each point is fixed at **-999.000**; the points with this elevation are ignored in the surface modeling operations.
- **Constant**: all the points have the same elevation, which is entered in the **Elevation** field.
- **Variable**: each point has its own elevation which can be edited in the **Z** column of the table of points.



The dialog box 'Entity Properties' shows the following settings:

- Layer: CTR-Topo
- Type: Polyline
- Elevation type:
  - ☐ Nil
  - ☒ Constant
  - ☐ Variable
- Elevation: 290.000
- ☒ Linked
- ☐ Closed
- ☐ Attribute by layer
- Nbr. points: 95
- Length: 1250.583
- Area: 56695.000

Num	X	Y	Z
84	394114.000	5613246.000	290.000
85	394101.000	5613228.000	290.000
86	394096.000	5613222.000	290.000
87	394083.000	5613201.000	290.000
88	394068.000	5613178.000	290.000
89	394051.000	5613155.000	290.000
90	394040.000	5613141.000	290.000

Figure 2-3 - Property window of an entity

#### 6.1.4. Closing a polyline

The checkbox labelled **Closed** is only active if the current cartographic element is a **Polyline** type. This box is used to define if the polyline will be closed or not.

Unlike a closed **Polyline** element, a **Polygon** element can have as an attribute the filling defined for the element or for the layer to which it belongs.

#### 6.1.5. Drawing attributes of the entity

When the box **Attribute by layer** is checked, the entity takes the same drawing attributes as the layer to which it belongs.

Note that if the attributes of a layer are modified (see paragraph 7.2), all the elements belonging to this layer and whose **Attribute by layer** box is checked will be modified.

If the box is not checked, the user can define the desired drawing attributes for the element by clicking on the attributes button labelled with the layer name. This button shows the current text, line and filling attributes for the entity. Clicking on this button opens the drawing attributes configuration window (see chapter 9 of the X3D guide).

#### 6.1.6. Table of points

The table at the bottom of the property window shows the X and Y coordinates and the elevation of the points of the cartographic element.

The row selected in the table corresponds to the point closest to the location where the double-click or the click with the right button was done to open the property window.

The user can edit the X and Y coordinates of the points and eventually their elevation, if the type is **Variable**.

The total number of points belonging to the current element is displayed in the **Nbr. points** field.

The length of the perimeter of the cartographic entity is displayed in the **Length** field. This value applies only to the polylines and polygons. The perimeter does not include the parts of an entity that are not linked.

The area of a polyline or polygon is also calculated and displayed in the **Area** field. In the case of an open polyline, the program closes the entity for the calculation.

### 6.2. Creating a cartographic entity



The **New Entity** option of the **Mapping** menu has 3 options: **Points**, **Polyline** and **Polygon**. By selecting one of the 3 options, the property window opens.

The **layer** displayed by default will be the layer of the last entity selected, or will be null if no entity was selected before. The user chooses the desired layer in the scrolling list or enters the name of a new layer.


The **type of the element** is by default the one of the selected option, but can be modified from the scrolling list.

The **type of elevation** can then be selected. With **Nil**, the **Elevation** field is set to **-999** and becomes inactive. With **Constant**, the user can enter the value for all the points in the **Elevation** field. With **Variable**, the **n/a** value is entered in the **Elevation** field and it becomes inactive. The user will define the elevation of each point of the entity during its creation.

For a new polyline, the user can check **Closed** to automatically close the polyline when it is created.

The user can check the **Attribute by layer** box. The entity will then take the attributes of the layer to which it belongs. Otherwise, the user can define the desired drawing attributes for the


element by clicking on the attributes button labeled with the layer name, which opens the drawing attributes window (see chapter 9 of the X3D guide).

When the characteristics of the entity to be created are defined, click **OK** and the cursor takes the format . For each point, each point of a polyline or each point of a polygon, place the cursor at the desired location and click with the left button of the mouse. A dotted line will follow the last point clicked in the case of polylines and polygons. If the elevation is **variable**, a box will open after each click to enter the elevation of the point. The elevation proposed by default is the one of the previously created point. To end the creation of the entity, click with the right button of the mouse to display a popup menu and select **Terminate**.

### 6.3. Copying a cartographic entity




The user can create a copy of a cartographic entity by selecting the **Copy Entity** option of the **Mapping** menu which becomes checked, or this button.

Select the entity to copy by clicking on it with the left button of the mouse. The cursor then takes the format . Click again on the entity and hold down the left button while dragging the copy of the entity to the desired location, and release. A dotted line follows the movement. You can create as many copies as needed, the option remaining active. To copy another entity, simply select it and repeat the steps.

### 6.4. Moving a cartographic entity



The user can move a cartographic entity by selecting the **Move Entity** option of the **Mapping** menu which becomes checked, or this button.

Select the entity to move by clicking on it with the left button of the mouse. The cursor then takes the format . Click again on the entity and hold down the left button while dragging the entity to the desired location, and release. A dotted line follows the movement. You can move the entity as many times as needed, the option remaining active. To move another entity, select it and repeat the steps.

### 6.5. Linking two cartographic entities




The user can link two cartographic entities by selecting the **Link Two Entities** option of the **Mapping** menu which becomes checked, or this button.

This option is under development.

### 6.6. Splitting a cartographic entity



The user can split a cartographic entity by selecting the **Split Entity** option of the **Mapping** menu which becomes checked, or this button.


Select the entity to split by clicking on it with the left button of the mouse. The cursor then takes the format . Then, click on the segment to delete. This operation cannot be done on an entity with less than 3 segments or on the end segments. You can split the entity as many times as needed, the option remaining active. To split another entity, select it and repeat the steps.

Note: if a polygon is split, the entity now split will remain of type "Polygon".

### 6.7. Deleting a cartographic entity




The user can delete a cartographic entity by selecting the **Delete Entity** option of the **Mapping** menu which becomes checked, or this button.

Select the entity to delete by clicking on it with the left button of the mouse. The cursor then takes this format + . Click again on the entity to delete it. You can delete as many entities as needed, the option remaining active. To delete another entity, simply select it and repeat the steps.

### 6.8. Adding a point



The user can add a point to an entity. First, select the entity to edit by clicking on it with the left button of the mouse. Then, select the **Add Point** option of the **Mapping** menu which becomes checked, or this button.


The cursor then takes this format + . A dotted line follows the cursor and shows the adjusted shape of the entity linking the two closest points. Left click to add the new point. If the point is added to an entity with variable elevation, the elevation of the point to its right (towards a greater point number) is given by default, but the user can modify it in the property window. To add another point, select **Add Point** again.

### 6.9. Moving a point



The user can move a point of an entity by selecting the **Move Point** option of the **Mapping** menu which becomes checked, or this button.

Select the entity containing a point to move by clicking on it with the left button of the mouse.


The cursor then takes this format + . Then, click on the point to move, hold down and drag the point to the desired location, and release. A dotted line follows the movement. You can move any point of the entity as many times as needed, the option remaining active. To move a point of another entity, select it and repeat the steps.

### 6.10. Deleting a point



The user can delete a point of an entity by selecting the **Delete Point** option of the **Mapping** menu which becomes checked, or this button.

Select the entity containing a point to delete by clicking on it with the left button of the mouse.

The cursor then takes this format + . Then, click on the point to delete. The shape of the entity is automatically adjusted. You can delete as many points of the entity as needed, the option remaining active. To delete a point of another entity, select it and repeat the steps.

### 6.11. Pop-up menu

By selecting an entity and clicking on it with the right button of the mouse, a popup menu is displayed, showing the options "Copy" to "Add Point" (paragraphs 6.3 to 6.8) as well as the Properties option (paragraph 6.1).



If the popup menu is called when the cursor is on a point of the entity, the options “Move Point” and “Delete Point” are also available (paragraphs 6.9 and 6.10). A message asks the user to confirm a point deletion.

## 7. MODELING MENU

The **Modeling** menu, shown in figure 2-4, is specific to the SM2 library. It allows:

- To prepare and edit cartographic maps,
- To prepare data terrain models by kriging or triangulation,
- To perform operations on the data terrain models,
- To calculate isovalue contours,
- To perform volume calculations.

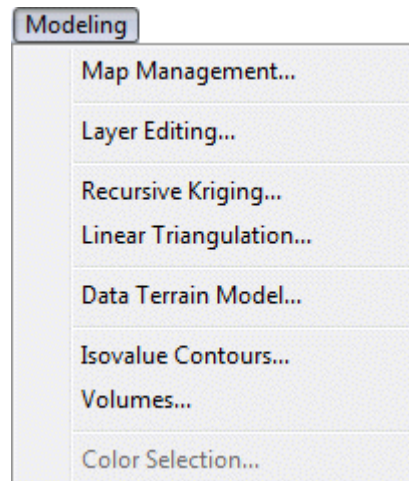


Figure 2-4 - Modeling menu

### 7.1. Map management



The **Map Management** option of the **Modeling** menu opens the map editing window shown in figure 2-5.

#### 7.1.1. Data file

The **File** field of the map editing window shows the name of the file whose content is displayed in the plan view. This file is opened via the **Open Data** option of the **File** menu. A file can also be opened by clicking on the button at the extreme right of the **File** field, which opens the Windows file selection window. The selection of a file replaces the content of the plan view and updates the fields of the map editing window. The **directory** of the current data file is indicated above.

#### 7.1.2. Import of a data file

If the data file is cartographic (all except the .dem, .dtm, .k2d, .tin, .grd, .stl and .igs files), it is possible to add a data file to the one already opened, by clicking on the **Import** button, which



opens the Windows file selection window. The selected file will be combined to the current file, which can then be saved with the **Save as** button at the bottom of the window.

### 7.1.3. Background file

A background file cannot be edited. It can be opened via the **New Data** option of the **File** menu (see chapter 2 of the X3D guide). Then, the **background** field of the map editing window will contain this file name.

The user can also open a background file, or even more than one successively, in the map editing window. Click on the “+” button, right of the **background** field, to open the Windows file selection window. When a background file is selected, it is added to the plan view.

If the domain of the selected background file is completely disjoint from the domain of the foreground file, a message warns the user and the file is not opened. If the domain of the selected background file is partially disjoint from the domain of the foreground file, a message warns the user but the file is still opened.

The **background** field shows the identification of the last selected file. The scrolling list associated to this field shows all the background files displayed in the plan view. Click on the “-” button to remove the selected background file. The plan view is then automatically adjusted.

### 7.1.4. File of polygonal limit

The user can add a file of polygonal limit in the plan view by clicking on the button at the extreme right of the **Polygonal limit** field, which opens the Windows file selection window for the **.lim** files. The selection of a file displays it on the plan view.

When a polygonal limit is displayed in the plan view, the user can clip the data displayed in the foreground inside or outside the limit.

The user selects the option **Delete points outside** or **inside** of the polygonal limit. Then, he clicks on the **Clip** button to purge the data either outside or inside the limit.

The checkbox **Add points to polylines intersecting clipping boundary** is used so that no data is lost in the clipping. Indeed, if the box is not checked, any polyline intersecting the polygonal limit will be deleted during the clipping. By checking the box, for each polyline intersecting the polygonal limit, a point is added at the intersection of the polyline and the limit, which allows deleting only the part of the polylines exceeding the limit.

### 7.1.5. File domain and additional information

The next section shows the characteristics of the file’s cartographic domain. The file domain is a rectangle defined by the coordinates of its **bottom left corner**, by its **width**, its **height** and the **angle** of its sides with the cardinal directions. In addition to the domain limits, there are elevation limits **Z max** and **Z min**. Generally, the file’s data are all contained in its domain, but SM2 allows defining a smaller domain, leaving some data outside the domain.

The **type** field indicates the type of file opened. The **version** field indicates with which version of SM2 the file was created. The reference system is not used.

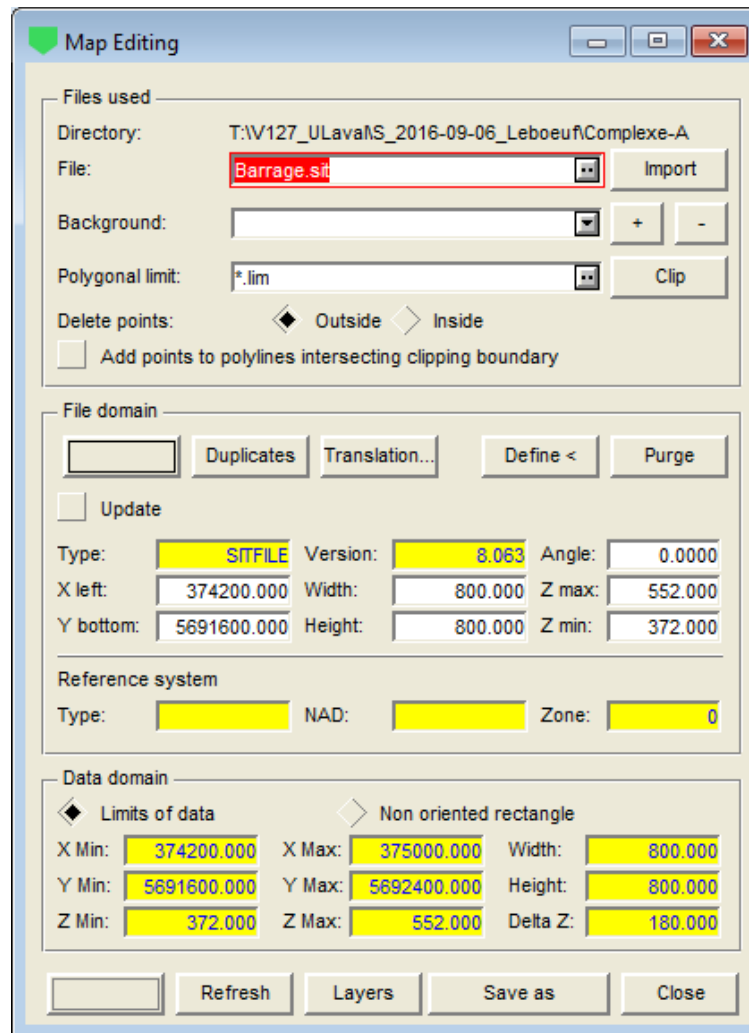


Figure 2-5 - Map editing window


### 7.1.6. Operations on the file domain

The **Update** box is used to automatically update the file domain depending on the operations done. For example, if the user imports a file, the file domain can be updated to take into account the new data. Similarly, after clipping around a polygonal limit, the file domain can be updated to include only the remaining data.

The **attributes** button is used to define the attributes of the limit of the domain defined with the **Define <** button, with the line and marker (for the corners of the domain) attributes.

The **Duplicates** button is used to eliminate identical entities in a file. Only the entities being exactly the same (same coordinates for their points) will be deleted.

The **Translation** button is in development.

By clicking on the **Define <** button, the map editing window goes in background and the cursor takes this format: . The user can then define a new rectangular domain in the plan view, by

clicking at one of the corners and dragging to the opposite corner, then releasing the left button of the mouse. The window then reappears, showing the new values for the file domain.

The **Purge** button is used to delete the data outside the defined file domain, in plan and elevation. The user defines the domain directly in the fields, or with the **Define <** button. The file can then be saved.

Please note that the option **Add points to polylines intersecting clipping boundary** is also used for the purge. Indeed, if the box is not checked, any polyline intersecting the domain limit will be deleted during the purge. By checking the box, for each polyline intersecting the domain limit, a point is added at the intersection of the polyline and the limit, which allows deleting only the part of the polylines exceeding the domain limit.

### 7.1.7. Data domain

The bottom of the map editing window shows the data domain - either the **limits of the data**, or the **non-oriented rectangle**, according to the radio button selected.

The limits cannot be edited. They change if the content of the file changes, in particular through a purge or a file import.

The **limits of the data** express the rectangle with no angle with the cardinal directions, and containing exactly the file data. This rectangle is defined by its extreme coordinates in X, Y and Z, its width, its height and its differential in elevation.

The **non-oriented rectangle** expresses the rectangle based on the cardinal directions, and containing exactly the oriented domain of the data.

If the angle of the domain is 0, the non-oriented rectangle and the data limits are the same.

### 7.1.8. Other operations

The **attributes** button is used to define the line (color and size) and the marker (color, size and period) used when selecting an entity in "Mapping" mode (see paragraph 6). The marker is automatically an empty square.

The **Refresh** button at the bottom of the map editing window updates the plan view with the last operations done, without exiting the editing window and without saving the data file.

The **Layers** button opens the layer editing window (see paragraph 7.2).

The **Save as** button opens the Window file saving window to save the data file with a new name. To save directly the data file, the user will use the **Save data** option of the **File** menu.

The **Close** button closes the editing window but does not save the data file.

## 7.2. Layer editing



The **Layer Editing** option of the **Modeling** menu opens the layer editing window shown in figure 2-6. The button shown here is equivalent. The window can also be opened via the **Layers** button of the map editing window.

This window shows the layers of cartographic elements contained in the data file (of **cartographic**, **AutoCAD** or **point** type (.sit, .ctr, .dxf, .xyz or .yxz)). These layers are generally present in AutoCAD .dxf files that can be opened with SM2. Remember that a layer can also be created during the creation of a series of points, a polyline or a polygon.

The field **Number of layers** shows the number of layers in the data file.

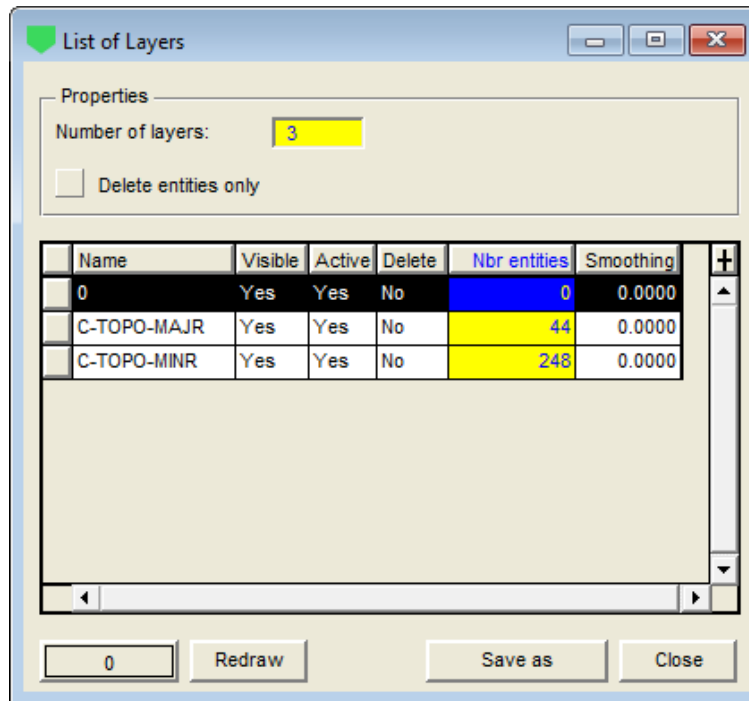


Figure 2-6 - Layer editing window

The table at the bottom of the window shows all the layers present in the data file. The content of the cells of the **Visible**, **Active** and **Delete** columns can be changed by double-clicking in the cell itself, or on the column's header to change the value for all the cells of the column.

- The **name** of the layer; it cannot be modified.
- The **Visible** column is used to display or hide a layer. By default, all the layers are **Visible**, with **Yes** in their cell.
- When a layer is **active**, the user can select its elements. By default, all the layers are **Active**, with **Yes** in their cell. A layer can be visible and inactive.
- Deleting a layer means deleting all the cartographic elements associated to that layer. To do this, put **Yes** in the **Delete** cell of the layer you wish to delete. The deletion is only saved when the **Save** button is clicked. Otherwise, the user can reshown the deleted layer by putting **No** in its cell.

If the box **Delete entities only** is checked, all the elements of a layer will be deleted but the layer itself will be kept, with no elements (0). On the contrary, if the box **Delete entities only** is not checked, all the elements of the layer, as well as the layer itself, will be deleted.

- The next column shows the **Number of entities** in each layer. These cells cannot be edited.

- The **smoothing** column is for future development.

The **attributes** button at the bottom left of the layer editing window is labeled with the name of the current layer. The user can modify the drawing attributes for the selected layer, including the line and the marker for the polylines, the points and the contours of the polygons, and the filling for the polygons. The drawing attributes only affect the elements that belong to the layer and whose box **Attribute by layer** in their property window is checked. See paragraph 6.1.5.

Clicking on the **Redraw** button updates the plan view with the last operations done on the layers, without exiting the editing window and without saving the data file.

The **Save as** button opens the Window file saving window. The user can give a new name or save over the previous file.

The **Close** button closes the editing window and updates the plan view with the last operations done on the layers but does not save the data file.

The **X** button closes the editing window and applies no changes in the plan view.

### 7.3. Recursive kriging

Once the site is defined adequately, the user can create the site terrain model, commonly called DTM for "Data Terrain Model". This model is a rectangular grid that covers the entire site and whose nodes have the terrain elevation. Conceptually, the constitution of a DTM is similar to placing a "net" on the site, consisting of rectangular meshes, and which moulds the terrain perfectly.

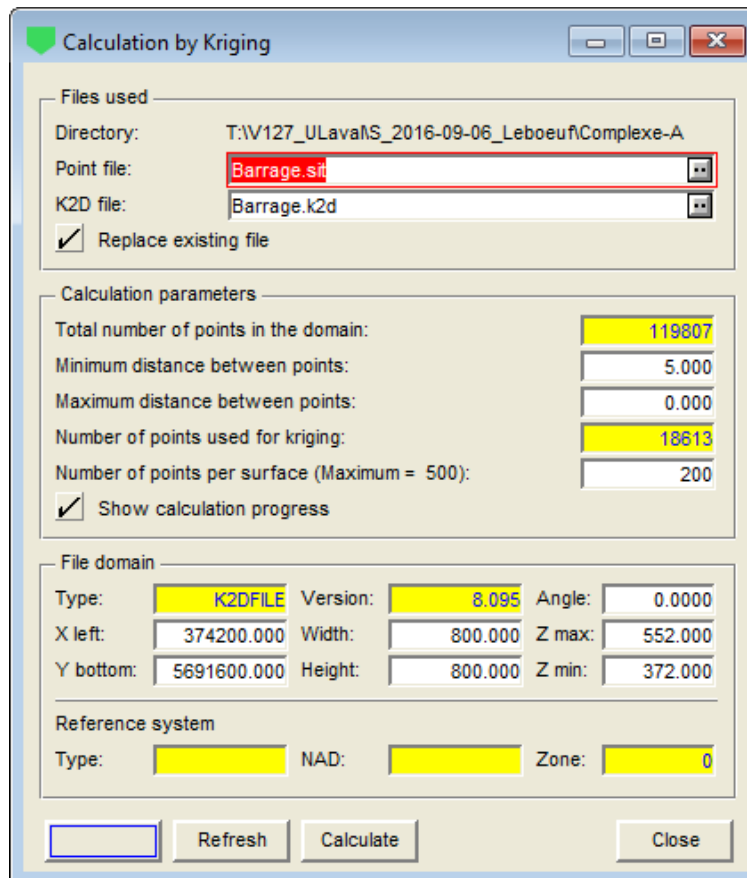
Before calculating the DTM, the user must establish the equation described by the terrain surface inside the site using the points that have an elevation. The first method offered by SM2 is the **recursive kriging**.



The **Recursive Kriging** option of the **Modeling** menu opens the calculation by kriging window shown in figure 2-7. The button shown here is equivalent.

In the **Files used** section:

- The **Point file** shows the name of the file whose content is displayed in the plan view. This file is opened via the **Open Data** option of the **File** menu. A file can also be opened by clicking on the button at the extreme right of the **Point file** field. The selection of a file replaces the content of the plan view and updates the fields of the calculation window. The file must be of **cartographic**, **AutoCAD** or **point** type (.sit, .ctr, .dxf, .xyz or .yxz) for the calculation by kriging.
- The **directory** of the current point file is indicated above.
- By default, the **K2D file** shows the same file name, with the .k2d extension. The user can modify it by clicking on the button at the extreme right of the field which opens the file saving window.
- The checkbox **Replace existing file** is used to overwrite an existing file, without a confirmation message being sent.



**Calculation by Kriging**

**Files used**

Directory: T:\V127\_ULava\NS\_2016-09-06\_Leboeuf\Complexe-A

Point file: **Barrage.sit**

K2D file: Barrage.k2d

☒ Replace existing file

**Calculation parameters**

Total number of points in the domain: 119807

Minimum distance between points: 5.000

Maximum distance between points: 0.000

Number of points used for kriging: 18613

Number of points per surface (Maximum = 500): 200

☒ Show calculation progress

**File domain**

Type: K2DFILE Version: 8.095 Angle: 0.0000

X left: 374200.000 Width: 800.000 Z max: 552.000

Y bottom: 5691600.000 Height: 800.000 Z min: 372.000

**Reference system**

Type: NAD: Zone: 0

Refresh Calculate Close

Figure 2-7 - Calculation by kriging

In the **Calculation parameters** section:

- The **total number of points in the domain** indicates the total number of points contained in the file domain. This information cannot be edited.
- The **minimum distance between points** can be modified. If points are located at a distance less than this minimum distance, they will not be used in the calculation.  
The smaller the minimum distance, the more precision the result of the calculation by kriging will have. However, increasing the precision also increases the calculation time; the value 0 gives the maximum precision. SM2 calculates the value that presents the best ratio "precision vs calculation time" which is proposed by default.
- The **maximum distance between points** can be modified. Points are then added to the line between two points located at a distance greater than the maximum distance, and the points are used in the calculation. SM2 proposes by default a maximum distance of 0, no adding any point.
- The **number of points used for kriging** indicates the number of points with elevation that will be used in the kriging operation. This value changes with the minimum and maximum distances between the points.
- The **number of points per surface** can be modified, with a maximum value of 500. This number is used to adjust the size of the kriging surfaces. The domain is divided in four

sections, then each can be re-divided in four until each section contains no more points than the maximum number of points indicated.

- By selecting the option **Show calculation progress**, the progressive display of the calculation will be shown. The display sometimes stops, but the calculation is still being done.

In the **File domain** section:

- The **type** field indicates the type of file that will be created (K2DFILE). The **version** field indicates with which version of SM2 the file will be created.
- The next fields define the domain of the file that will be created. The domain is a rectangle defined by the coordinates of its **bottom left corner**, by its **width**, its **height**, by the **angle** of its sides with the cardinal directions and by its elevation limits **Z max** and **Z min**. The user can modify the domain of the k2d file to be created, but not its elevation limits.
- The reference system is not used.

The **attributes** button is used to define a marker, its dimension and its color to identify the points that will be used in the calculation by kriging, when clicking on the **Refresh** button.

The **Refresh** button updates the plan view and shows the points that will be used in the calculation by kriging.

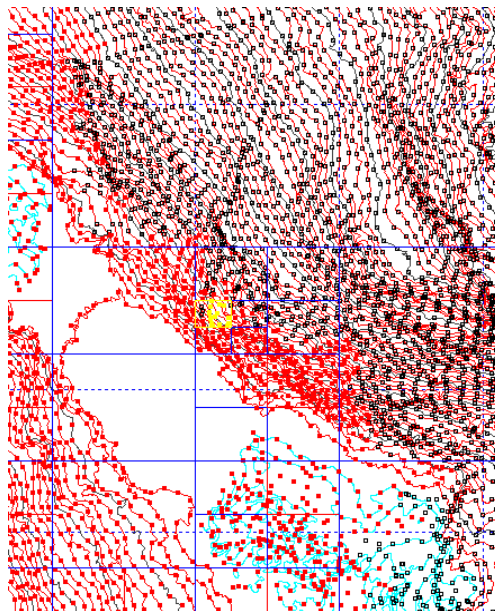


Figure 2-8 - Plan view during kriging

The **Calculate** button starts the kriging. The calculation by kriging window reappears once the calculation is done. Figure 2-8 shows the display of the plan view during the kriging. The red points have already been kriged, the yellow points are being calculated, and the black points will then be calculated.

The **Close** button closes the window.



## 7.4. Linear Triangulation

As described at the beginning of paragraph 7.3, before calculating the DTM, the user must establish the equation described by the terrain surface inside the site using the points that have an elevation. The second method offered by SM2 is the **linear triangulation**.



The **Linear Triangulation** option of the **Modeling** menu opens the calculation by triangulation window shown in figure 2-9. The button shown here is equivalent.

In the **Files used** section:

- The **Point file** shows the name of the file whose content is displayed in the plan view. This file is opened via the **Open Data** option of the **File** menu. A file can also be opened by clicking on the button at the extreme right of the **Point file** field. The selection of a file replaces the content of the plan view and updates the fields of the calculation window. The file must be of **cartographic**, **AutoCAD** or **point** type (.sit, .ctr, .dxf, .xyz or .yxz) for the calculation by triangulation.
- The **directory** of the current point file is indicated above.
- The **limit file** is for future development.
- By default, the **TIN file** shows the same file name, with the .tin extension. The user can modify it by clicking on the button at the extreme right of the field which opens the file saving window.
- The checkbox **Replace existing file** is used to overwrite an existing file, without a confirmation message being sent.

In the **Calculation parameters** section:

- The **total number of points in the domain** indicates the total number of points contained in the file domain. This information cannot be edited.
- The **minimum distance between points** can be modified. If points are located at a distance less than this minimum distance, they will not be used in the calculation.

The smaller the minimum distance, the more precision the result of the calculation by triangulation will have. However, increasing the precision also increases the calculation time; the value 0 gives the maximum precision. SM2 calculates the value that presents the best ratio "precision vs calculation time" which is proposed by default.

- The **maximum distance between points** can be modified. Points are then added to the line between two points located at a distance greater than the maximum distance, and the points are used in the calculation. SM2 proposes by default a maximum distance of 0, no adding any point.
- The **number of points selected for triangulation** indicates the number of points with elevation that will be used in the triangulation operation. This value changes with the minimum and maximum distances between the points.
- By selecting the option **Add points to cover the whole domain**, a point will be added at each corner of the domain to be able to calculate the elevation of all the points.

In the **File domain** section:



- The **type** field indicates the type of file that will be created (TINFILE). The **version** field indicates with which version of SM2 the file will be created.
- The next fields define the domain of the file that will be created. The domain is a rectangle defined by the coordinates of its **bottom left corner**, by its **width**, its **height**, by the **angle** of its sides with the cardinal directions and by its elevation limits **Z max** and **Z min**. The user can modify the domain of the .tin file to be created, but not its elevation limits.
- The reference system is not used.

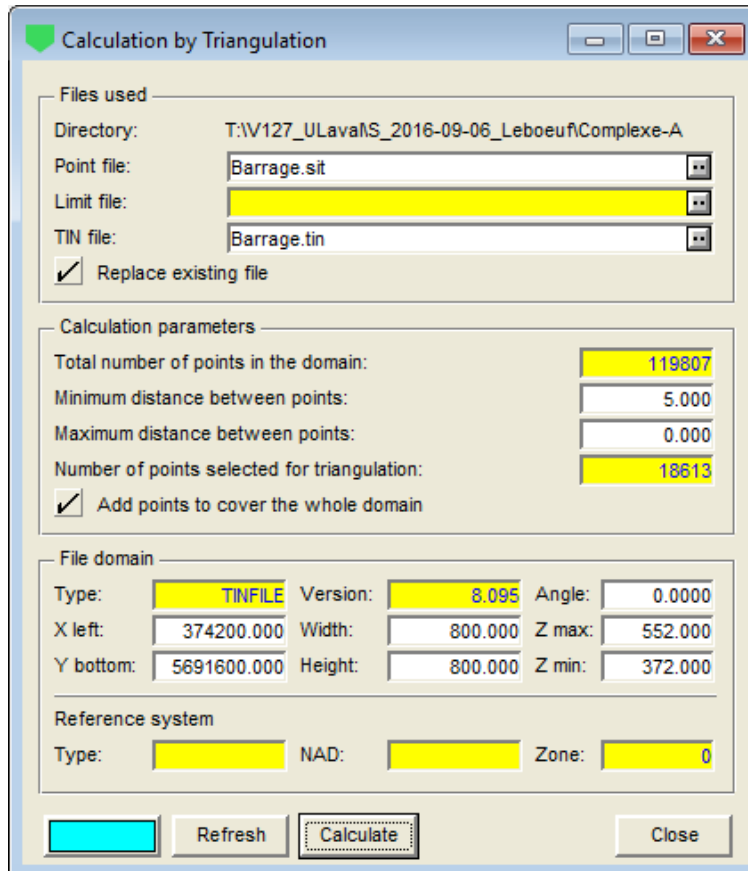


Figure 2-9 - Calculation by triangulation

The **attributes** button is used to define a marker, its dimension and its color to identify the points that will be used in the calculation by triangulation, when clicking on the **Refresh** button. The line and filling attributes will be those used to show the result of the calculation by triangulation.

The **Refresh** button updates the plan view and shows the points that will be used in the calculation by triangulation.

The **Calculate** button starts the calculation by triangulation. Once the calculation is done, the display shows the result with the attributes defined. Figure 2-10 shows the display of the plan view before and after the triangulation.

The **Close** button closes the window.

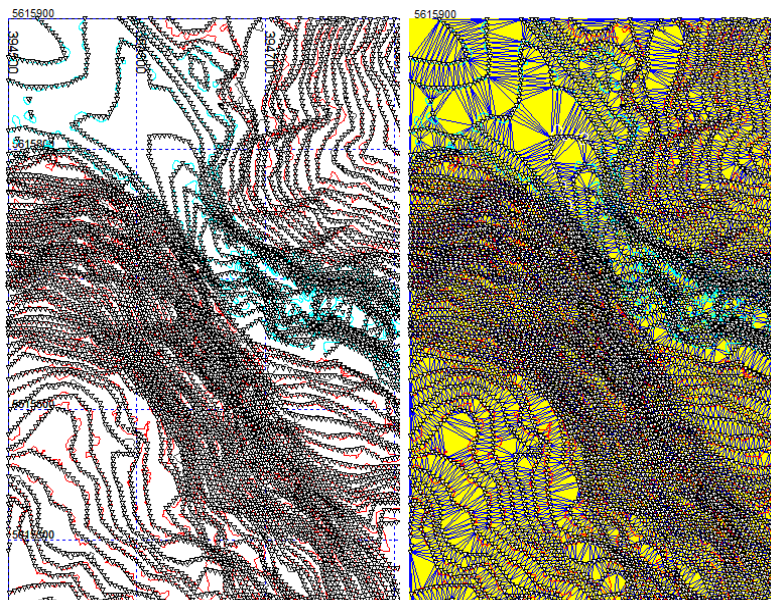


Figure 2-10 - Plan view before and after triangulation

## 7.5. Data Terrain Model



All the operations on terrain models (DTM for Data Terrain Model) are done from the operations on DTM window, opened via the **Data Terrain Model** option of the **Modeling** menu or the button shown here.

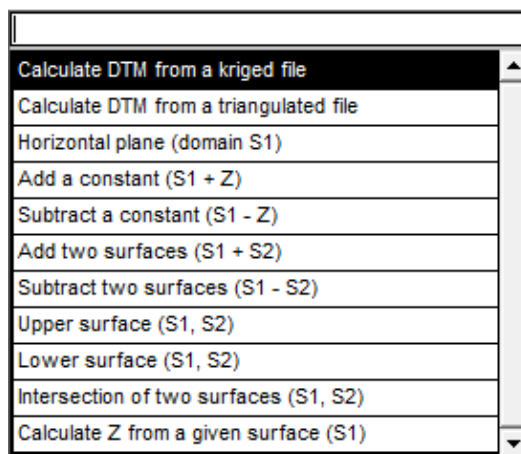


Figure 2-11 - List of operations on the DTM

The scrolling list associated to the **Operation** field is shown in figure 2-11. Each operation will be described in the following paragraphs.

### 7.5.1. Calculate the DTM from a kriged file

In the **Files used** section:

- If a .k2d file was opened in the plan view via the option **Open Data** of the **File** menu, the operation **Calculate the DTM from a kriged file** will be automatically selected in the Operations on DTM window, as shown in figure 2-12. A file can also be opened by clicking

on the button at the extreme right of the **Source file** field. The selection of a file replaces the content of the plan view and updates the fields of the window. The file must be of **.k2d** type for this operation.

- The **directory** of the source file is indicated above.
- By default, the **Resulting DTM** shows the same file name, with the **.dtm** extension. The user can modify it by clicking on the button at the extreme right of the field, which opens the file saving window. The resulting files can be **.dtm** or **.grd** types.
- The checkbox **Replace existing file** is used to overwrite an existing file, without a confirmation message being sent.

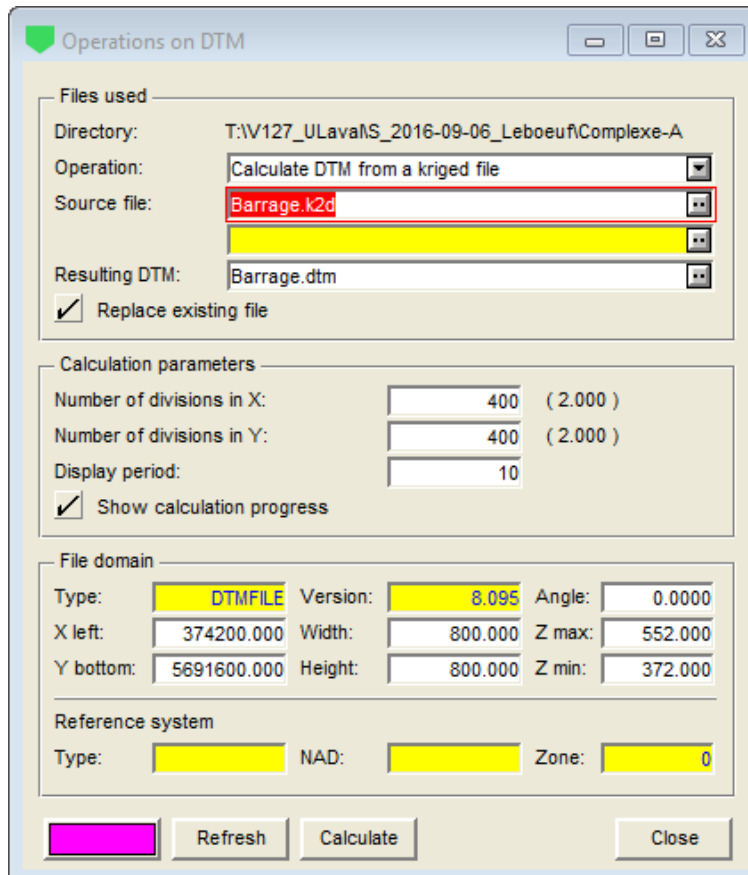


Figure 2-12 - Calculate the DTM from a kriged file

In the **Calculation parameters** section:

- the user specifies the **number of divisions in X and in Y** for the DTM calculation. The number in parentheses is the size of a cell. The maximum number of divisions is 1000 by 1000.
- The **display period** represents the period of the grids displayed in the DTM created, in 2D or 3D, as well as during the calculation, if the box **Show calculation progress** is checked. For example, a period of 10 will show 1 of 10 limits, in both directions.

In the **File domain** section:

- The **type** field indicates the type of file that will be created (DTMFILE). The **version** field indicates with which version of SM2 the file will be created.
- The next fields define the domain of the file that will be created. The domain is a rectangle defined by the coordinates of its **bottom left corner**, by its **width**, its **height**, by the **angle** of its sides with the cardinal directions and by its elevation limits **Z max** and **Z min**. The user can modify the domain of the .dtm file to be created, but not its elevation limits.
- The reference system is not used.

The **attributes** button is used to define the color of the grids as well as a filling, used when the DTM is displayed in OpenGL mode and the option "file color" is selected (see paragraph 7.8).

The **Refresh** button updates the plan view.

The **Calculate** button starts the calculation of the DTM from the kriged file.

The **Close** button closes the window.

### 7.5.2. Calculate the DTM from a triangulated file

If a .tin file is opened in the plan view via the option **Open Data** of the **File** menu, the operation **Calculate the DTM from a triangulated file** will be automatically selected in the operations on DTM window. A file can also be opened by clicking on the button at the extreme right of the **Source file** field, which opens the Windows file selection window. The selection of a file replaces the content of the plan view and updates the fields of the window. The file must be of .tin type for this operation.

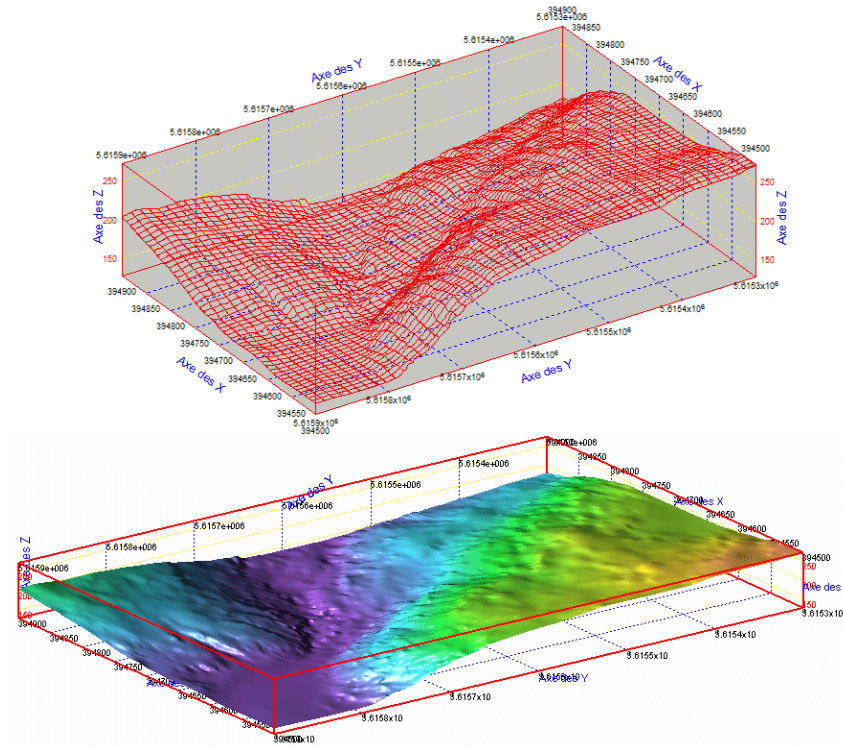


Figure 2-13 - Example of a calculated DTM in 3D and in OpenGL mode

All the other fields have the same function as for the calculation of the DTM from a kriged file. See paragraph 7.5.1. Figure 2-13 shows the result of a calculated DTM in 3D view and in OpenGL mode with the selection of elevation color (see paragraph 7.8).

### 7.5.3. Horizontal plane (domain S1)

If a .dtm file is opened in the plan view via the option **Open Data** of the **File** menu, the last operation done on a DTM will be selected by default in the operations on DTM window. The operation can be modified via the scrolling list.

With **horizontal plane (domain S1)**, as shown in figure 2-14, a plane surface is created by giving the same elevation to each node of a grid. The domain of this plane is the same as the domain of the reference DTM, except the Z values.

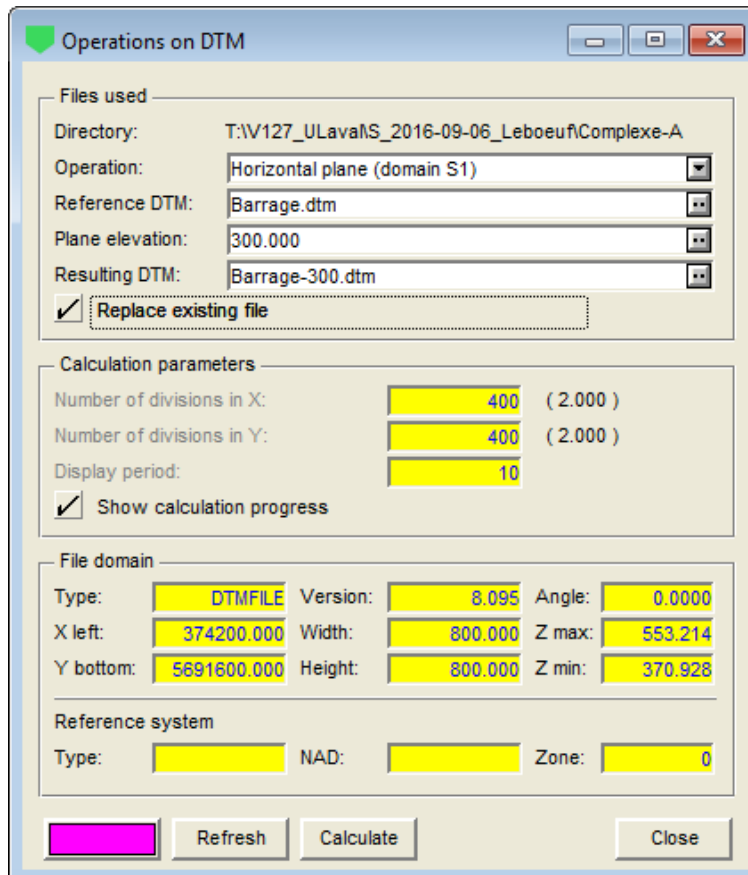


Figure 2-14 - Create a horizontal plane (domain S1)

The **Reference DTM** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of the **Reference DTM** field. This file must be of **.dtm** or **.grd** type for this operation. The file domain is updated when a reference file is selected.

The user then enters the horizontal **plane elevation** for all the nodes of the grid to create.

In the **Resulting DTM** field, the user directly enters the name in the field or clicks on the button at the extreme right of the field to open the file saving window.



The **file domain** cannot be modified, since the grid of the new file must be the same as the grid of the reference file.

The **attributes** button is used to define the color of the grids as well as a filling, used when the DTM is displayed in OpenGL mode and the option “file color” is selected (see paragraph 7.8).

Click **Calculate**. The operation is done almost instantaneously.

#### 7.5.4. Add a constant (S1 + Z)

With the **addition of a constant (S1 + Z)**, as shown in figure 2-15, a grid parallel to the original grid is created. The nodes of the grid created keep the same coordinates as the original grid, but the elevation of each node is increased by a constant value.

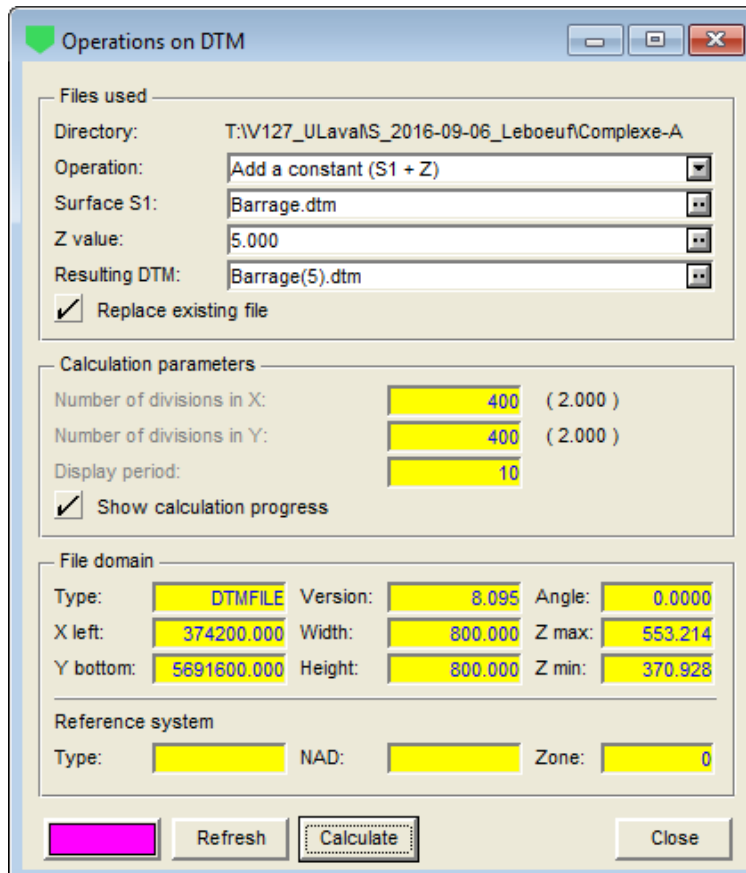


Figure 2-15 - Add a constant (S1 + Z)

The **Surface S1** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of **surface S1** field. This file must be of **.dtm** or **.grd** type for this operation. The file domain is updated when a S1 reference file is selected.

The user then enters the **Z value** to add.

In the **Resulting DTM** field, the user directly enters the name in the field or clicks on the button at the extreme right of the field to open the file saving window.

The **attributes** button is used to define the color of the grids as well as a filling, used when the DTM is displayed in OpenGL mode and the option “file color” is selected (see paragraph 7.8).

Click **Calculate**. The operation is done almost instantaneously.

#### 7.5.5. Subtract a constant (S1 - Z)

The **subtraction of a constant (S1 - Z)** is exactly like the addition of a constant, except that the Z value is subtracted instead of added to the elevation of each point of the original grid. See paragraph 7.5.4.

#### 7.5.6. Add two surfaces (S1 + S2)

The **addition of two surfaces (S1 + S2)** creates a surface by adding the elevation of each node from the first grid to the elevation of the corresponding node in the second grid. The domains and grids of the two files must be identical.

The **surface S1** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of **surface S1** field. This file must be of **.dtm** or **.grd** type for this operation. The file domain is updated when a S1 file is selected. The **surface S2** is selected the same way.

In the **Resulting DTM** field, the user directly enters the name in the field or clicks on the button at the extreme right of the field to open the file saving window.

The **file domain** cannot be modified, since the resulting file must have the same domain and grid as the S1 and S2 surfaces.

The **attributes** button is used to define the color of the grids as well as a filling, used when the DTM is displayed in OpenGL mode and the option “file color” is selected (see paragraph 7.8).

Click **Calculate**. The operation is done almost instantaneously. If the S1 and S2 surfaces don't have the same domain or same grid, a message warns the user and the operation is not done.

#### 7.5.7. Subtract two surfaces (S1 - S2)

The **subtraction of two surfaces (S1 - S2)** is exactly like the addition of two surfaces, except that it creates a surface by subtracting the elevation of each node from the second grid from the elevation of the corresponding node in the first grid. See paragraph 7.5.6.

#### 7.5.8. Upper surface (S1, S2)

With **upper surface (S1, S2)**, a surface is defined from the upper section of two grids. For each node, the highest elevation from the two surfaces is used for the new surface. The domains and grids of the two files must be identical.

The **surface S1** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of **surface S1** field. This file must be of **.dtm** or **.grd** type for this operation. The file domain is updated when a S1 file is selected. The **surface S2** is selected the same way.

In the **Resulting DTM** field, the user directly enters the name in the field or clicks on the button at the extreme right of the field to open the file saving window.

The **file domain** cannot be modified, since the resulting file will have the same domain and grid as the S1 and S2 surfaces.

The **attributes** button is used to define the color of the grids as well as a filling, used when the DTM is displayed in OpenGL mode and the option “file color” is selected (see paragraph 7.8).

Click **Calculate**. The operation is done almost instantaneously. If the S1 and S2 surfaces don't have the same domain or same grid, a message warns the user and the operation is not done.

#### 7.5.9. Lower surface (S1, S2)

The **lower surface (S1, S2)** is exactly like the upper surface of S1 and S2, except that it creates a surface representing the bottom section of the two grids, by using for each node the lowest elevation from the two surfaces. See paragraph 7.5.8.

#### 7.5.10. Intersection of two surfaces (S1, S2)

The **intersection of two surfaces (S1, S2)** calculates the intersection of two grids. The domains and grids of the two files must be identical. The result is a cartographic file with the **.int** extension.

The **surface S1** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of **surface S1** field. This file must be of **.dtm** or **.grd** type for this operation. The file domain is updated when a S1 file is selected. The **surface S2** is selected the same way.

The **Resulting file** shows the same file name as S1, with the **.int** extension. The user can modify it directly in the field or by clicking on the button at the extreme right of the field to open the file saving window.

The **file domain** cannot be modified, since the intersection file will have the same domain and grid as the S1 and S2 surfaces.

The **attributes** button is not used.

Click **Calculate**. The operation is done almost instantaneously. If the S1 and S2 surfaces don't have the same domain or same grid, a message warns the user and the operation is not done.

#### 7.5.11. Calculate Z from a given surface (S1)

The **calculation of Z from a given surface (S1)** calculates the elevation of the points constituting the elements of a cartographic file from a DTM.

The **Source file** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of the **Source file** field, which opens the Windows file selection window. This file must be of **.sit** type for this operation. The file domain is updated when a source file is selected.

The **Reference DTM** is then selected the same way. This file is the reference for the elevations.



In the **Resulting DTM** field, the user directly enters the name in the field or clicks on the button at the extreme right of the field to open the file saving window. The extension is **.sit**.

Click **Calculate**.

## 7.6. Isovalue Contours

SM2 can be used to calculate from a DTM grid the isovalue curves (contours), i.e. the lines of equal elevation. SM2 creates a **.ctr** extension file.



The **Isovalue Contours** option of the **Modeling** menu opens the window for the calculation of isovalue curves, as shown in figure 2-16. The button shown here is equivalent.

In the **Files used** section:

- The **DTM file** field of the contours window shows the name of the file whose content is displayed in the plan view. This file is opened via the **Open Data** option of the **File** menu. A file can also be opened by clicking on the button at the extreme right of the **DTM file** field. The selection of a file replaces the content of the plan view and updates the fields of the calculation window. The file must be of **.dtm** or **.grd** type for this operation.
- The **directory** of the current DTM file is indicated above.
- By default, the **CTR file** field shows the same file name, with the **.ctr** extension. The user can modify it by clicking on the button at the extreme right of the field which opens the file saving window.
- The checkbox **Replace existing file** is used to overwrite an existing file, without a confirmation message being sent.

In the **Calculation parameters** section:

- The **distance between labels** is the distance in geographic units (meters) between the labels showing the elevation of each isovalue curve, if the labels are displayed. If a curve is shorter than the distance between labels indicated, only one label will be displayed in the middle of the curve.
- The **contour interval** for the intermediate curves is then indicated, in geographic units (meters). A curve will be displayed at each elevation interval indicated, from the minimum to the maximum elevation. A value is proposed by default, which takes into account the difference between the minimum and maximum elevations at the nodes of the DTM grid and the display dimensions.

The **attributes** button right of the contour interval field is used to modify the drawing attributes of these intermediate curves. The line as well as the text can be modified. Note that the height of the text is in geographic units (meters).

The **Labels** checkbox is used to display or not the labels of these intermediate curves.

- The **master contour interval** is then indicated, in geographic units (meters). A curve will be displayed at each elevation interval indicated, from the minimum to the maximum elevation. A value is proposed by default, which is 5 times the value of the contour interval for the intermediate curves.

The **attributes** button has the same role as for the intermediate curves.

The **Labels** checkbox is used to display or not the labels of the master curves.

- The values of the **minimum** and **maximum levels** of the DTM file covering the difference in elevation and taking into account the interval between the intermediate curves are displayed in the inactive fields. The user can change the **minimum** and **maximum levels** to use in the calculation.

If the values entered are outside the minimum and maximum elevations of the nodes of the grid, it is the values proposed that will be used in the calculation.

- In **additional contours**, the user can add the elevation of the curves to add in the calculation of the isovalue curves.

The **attributes** button right of additional contours is used to define the drawing attributes of the additional curves at a specified elevation.

- By selection the option **Show calculation progress**, the progressive display of the calculation will be shown.
- The **file domain** cannot be modified. It will be the same as the DTM file.

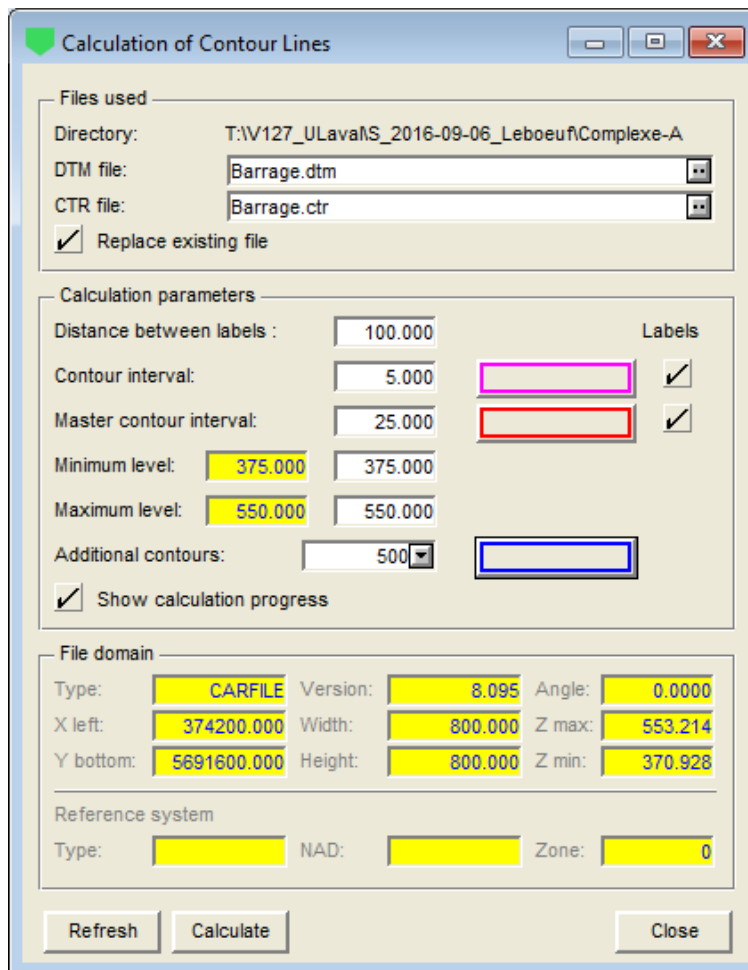


Figure 2-16 - Window for the calculation of contour lines

The **Calculate** button starts the calculation of the isovalue contours. The file produced is a cartographic file of .ctr type where the **CTR\_PRI** layer groups the master curves, the **CTR\_SEC**

layer groups the intermediate curves, and the **CTR\_USR** the additional curves. Figure 2-17 shows an example of a contours file in 3D.

The **Refresh** button updates the plan view. The **Close** button closes the window.

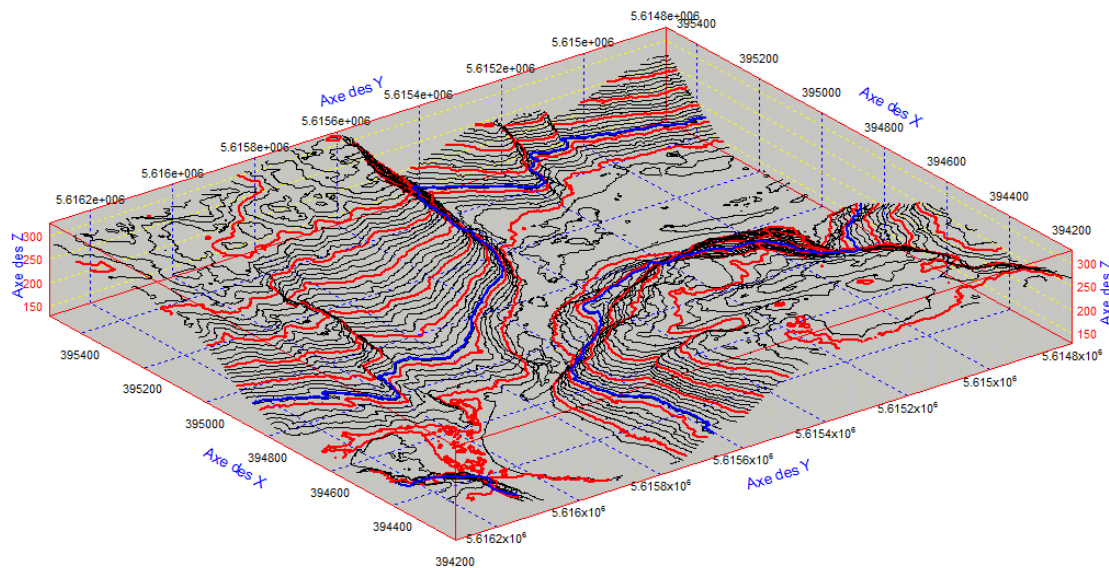


Figure 2-17 - Example of an isovalue contours file in 3D

## 7.7. Volumes

SM2 is used to calculate the volume between two surfaces, as well as the volume between a surface and planes of constant elevation. The calculated volumes are saved in a **.vol** text file.



The **Volumes** option of the **Modeling** menu opens the window for the calculation of volumes. The button shown here is equivalent. The two types of calculation are described below.

### 7.7.1. Volumes between two surfaces

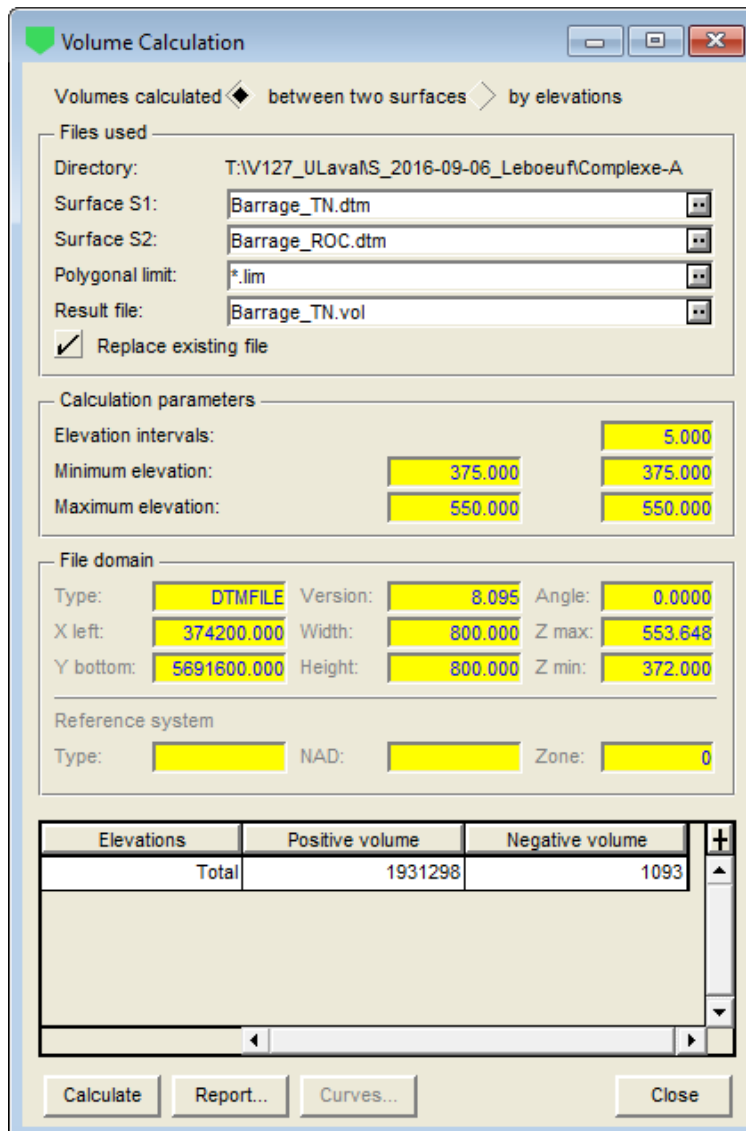
The **calculation of volumes between two surfaces** determines the positive and negative volumes between two surfaces. The volume is **positive** where the first selected surface has an elevation greater than the one of the second surface; the volume is said to be **negative** in the opposite case.

- The **surface S1** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of **surface S1** field. This file must be a **data terrain model** (.dtm, .ftm, .xtm) or a **grid** (.grd) for this operation. The **directory** of the source file is indicated above.
- The **surface S2** is selected the same way as the surface S1.
- The limit of the domain for the volume calculation is not restricted and can be defined by a polygon. The limits of the domain in which we want to calculate the volumes are contained in a **.lim** file. If no **.lim** file is selected, the program calculates the volumes over the entire domain of the selected surfaces.

If desired, select the **polygonal limit**. The .lim limits file can contain one or several polygons. If it contains several polygons, the first one defined will be the only one used in the volume calculation.

The polygon used can be partially or entirely outside the domain of the DTMs. If the polygon used from the limits file is completely outside the domain of the DTMs, the resulting volumes will be zero. If the polygon is partially outside the domain of the DTMs, the calculation will be done by taking into consideration only the portion of the polygon that is inside the domain of the DTMs.

- By default, the **Result file** field shows the same file name as S1, with the **.vol** extension. The user can modify it by clicking on the button at the extreme right of the field which opens the file saving window. The result files are of **text** type, the **.vol** extension being generally used to indicate the volume reports.
- The checkbox **Replace existing file** is used to overwrite an existing file, without a confirmation message being sent.



Volume Calculation

Volumes calculated ☒ between two surfaces ☐ by elevations

Files used

Directory: T:\V127\_ULava\IS\_2016-09-06\_Leboeuf\Complexe-A

Surface S1: Barrage\_TN.dtm

Surface S2: Barrage\_ROC.dtm

Polygonal limit: \*.lim

Result file: Barrage\_TN.vol

☒ Replace existing file

Calculation parameters

Elevation intervals: 5.000

Minimum elevation: 375.000 375.000

Maximum elevation: 550.000 550.000

File domain

Type: DTMFILE Version: 8.095 Angle: 0.0000

X left: 374200.000 Width: 800.000 Z max: 553.648

Y bottom: 5691600.000 Height: 800.000 Z min: 372.000

Reference system

Type: NAD: Zone: 0

Elevations	Positive volume	Negative volume
Total	1931298	1093

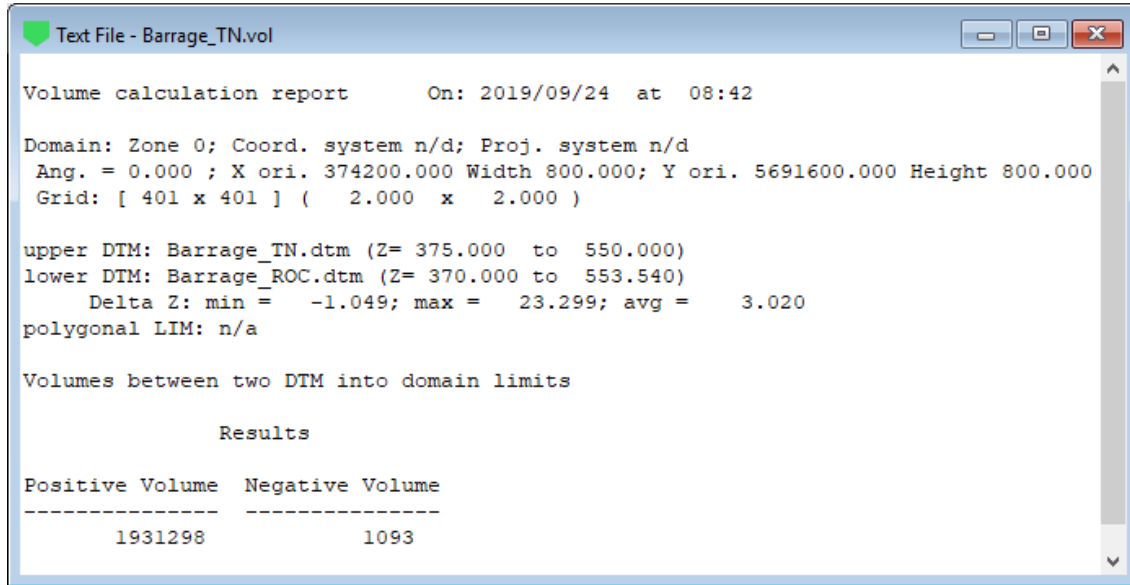
Calculate Report... Curves... Close

Figure 2-18 - Calculation of volumes between two surfaces

The calculation parameters and file domain have no effect for this type of calculation. They show the minimum and maximum elevations of the S1 file, as well as the domain of the S1 file.

The **Calculate** button starts the volume calculation. The positive and negative volumes are then displayed in the table at the bottom of the calculation window.

The **Report** button opens the **.vol** results file. An example for the above calculation is displayed in figure 2-19.



```

Volume calculation report      On: 2019/09/24 at 08:42

Domain: Zone 0; Coord. system n/d; Proj. system n/d
Ang. = 0.000 ; X ori. 374200.000 Width 800.000; Y ori. 5691600.000 Height 800.000
Grid: [ 401 x 401 ] ( 2.000 x 2.000 )

upper DTM: Barrage_TN.dtm (Z= 375.000 to 550.000)
lower DTM: Barrage_ROC.dtm (Z= 370.000 to 553.540)
Delta Z: min = -1.049; max = 23.299; avg = 3.020
polygonal LIM: n/a

Volumes between two DTM into domain limits

Results

Positive Volume  Negative Volume
-----
1931298          1093
  
```

Figure 2-19 - Calculation report of volumes between two surfaces

### 7.7.2. Volumes by elevations

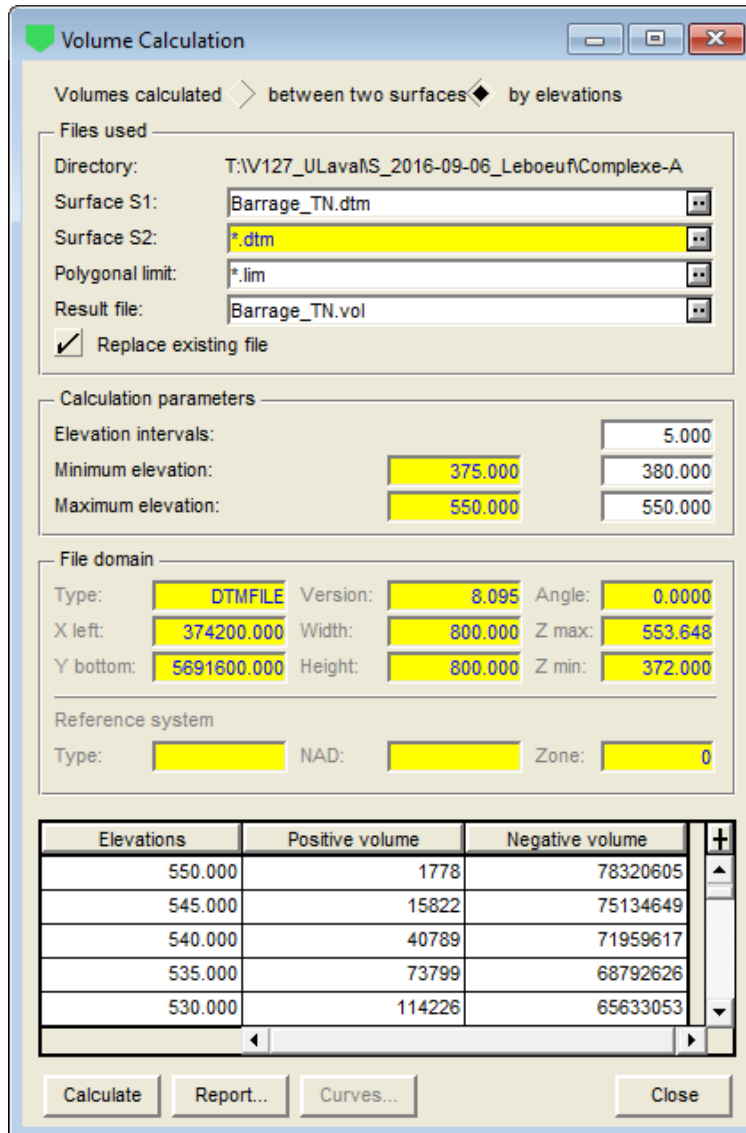
The **calculation of volumes by elevations** determines the positive and negative volumes between a surface and planes of constant elevation. The volume is **positive** where the selected surface has an elevation greater than the one of the plane; the volume is **negative** in the opposite case.

- The **surface S1** is either the file opened in the plan view, or another can be opened by clicking on the button at the extreme right of **surface S1** field. This file must be a **data terrain model** (.dtm, .ftm, .xtm) or a **grid** (.grd) for this operation. The **directory** of the source file is indicated above.
- The limit of the domain for the volume calculation is not restricted and can be defined by a polygon. The limits of the domain in which we want to calculate the volumes are contained in a **.lim** file. If no **.lim** file is selected, the program calculates the volumes over the entire domain of the selected surface.

Select the **polygonal limit**. The **.lim** limits file can contain one or several polygons. If it contains several polygons, the first one defined will be the only one used in the volume calculation.

The polygon used can be partially or entirely outside the domain of the DTMs. If the polygon used from the limits file is completely outside the domain of the DTMs, the resulting volumes will be zero. If the polygon is partially outside the domain of the DTMs, the calculation will be done by taking into consideration only the portion of the polygon that is inside the domain of the DTMs.

- By default, the **Result file** field shows the same file name as S1, with the **.vol** extension. The user can modify it by clicking on the button at the extreme right of the field which opens the file saving window. The result files are of **text** type, the **.vol** extension being generally used to indicate the volume reports.
- The checkbox **Replace existing file** is used to overwrite an existing file, without a confirmation message being sent.



**Volume Calculation**

Volumes calculated  $\triangleright$  between two surfaces  $\blacklozenge$  by elevations

**Files used**

Directory: T:\V127\_ULavaIS\_2016-09-06\_LeboeufComplexe-A

Surface S1: Barrage\_TN.dtm

Surface S2: \*.dtm

Polygonal limit: \*.lim

Result file: Barrage\_TN.vol

☒ Replace existing file

**Calculation parameters**

Elevation intervals: 5.000

Minimum elevation: 375.000 380.000

Maximum elevation: 550.000 550.000

**File domain**

Type: DTMFILE Version: 8.095 Angle: 0.0000

X left: 374200.000 Width: 800.000 Z max: 553.648

Y bottom: 5691600.000 Height: 800.000 Z min: 372.000

**Reference system**

Type: NAD: Zone: 0

Elevations	Positive volume	Negative volume
550.000	1778	78320605
545.000	15822	75134649
540.000	40789	71959617
535.000	73799	68792626
530.000	114226	65633053

Calculate Report... Curves... Close

Figure 2-20 - Calculation of volumes by elevations

- The **calculation parameters** show the **minimum and maximum elevations** of the S1 file according to the elevation interval.

The user can modify the **elevation intervals**. The volume calculations will be done for each interval.

The user can enter the **minimum and maximum elevations** of the planes for the volume calculations.

- The domain of the S1 file is displayed as information only.

The **Calculate** button starts the volume calculation. The positive and negative volumes are then displayed in the table at the bottom of the calculation window, for all the elevations calculated from the maximum to the minimum elevation.

The **Report** button opens the .vol results file. An example for the above calculation is displayed in figure 2-21.

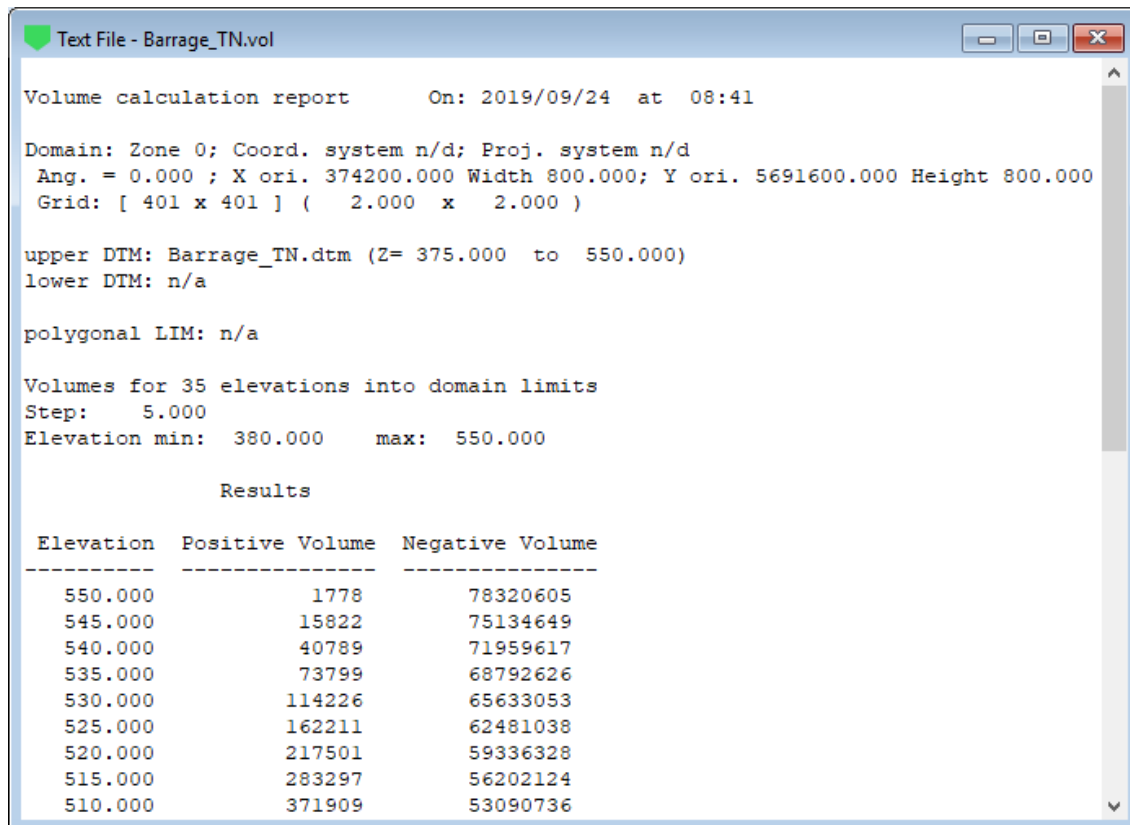


Figure 2-21 - Calculation report of volumes by elevations

## 7.8. Color Selection



The **Color Selection** option of the **Modeling** menu opens the color preferences window illustrated in figure 2-22. The button shown here is equivalent. This option is active only when SM2 is in OpenGL mode. See paragraph 3.

### 7.8.1. Color file

The information in the color editing window is saved in a .clr color file. Before opening the color editing window, the user can change the current file by selecting a file in the **Recent Colors** option of the **File** menu.

In the color editor, a colors file can be selected from the scrolling list associated to the top field. This list shows the last 9 colors files used. The user can also change the colors file by clicking on **Browse**, which opens the standard window for file selection on the directory of the current file. The proposed extension is .clr.



### 7.8.2. Color selection by elevation

The left section of the window is used to define the colors for the different elevations, with up to 10 elevations being defined. Each of the 10 **color buttons** opens a Windows color window where the user can select the desired color. The left bar shows the result of the color gradient.

The field right of each color button indicates the **elevation** corresponding to the color defined. The **checkbox** right of each elevation indicates if the color for the corresponding elevation will be used or not for the data file.

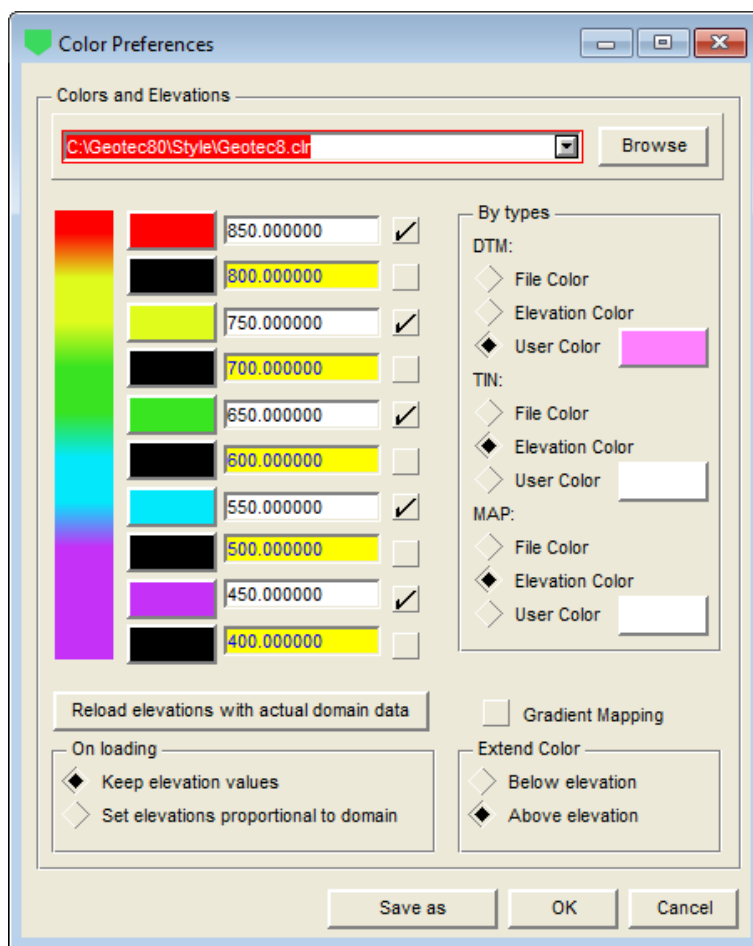


Figure 2-22 - Color Preferences window

### 7.8.3. Application of colors

The right section of the window is used to define the application of the colors by file type. For **DTM** files (including the **.dtm**, **.ftm**, **.xtm** and **.grd**) and **TIN** files, the user has three choices. The MAP files are not used in the current version.

- **File Color** will use the filling color of the data file. This color is defined when the data file is created.
- **Elevation Color** will use the colors by elevation defined in the current color file.
- **User Color** allows the user to define another color, by clicking on the button right of the label.



---

#### 7.8.4. Other parameters

In the **On loading** rectangle:

- The user can decide to **keep the elevation values**, meaning that the value defined for each color does not change, even if the minimum and maximum elevations in the data file are different from the values indicated.

With this option, the user will still be able to **reload the elevations with actual domain data** by clicking on the corresponding button.

- With the option **Set elevations proportional to domain**, the elevations for each color are adjusted based on the elevations of the data file, to always use ten (10) values.

When the **Gradient Mapping** option is selected, the color is displayed for the corresponding exact elevation, and a color gradient is added between the different elevations. If the **Gradient Mapping** option is not selected, the user can choose to extend colors:

- **Below elevation:** The color is used for the surfaces whose corresponding elevation is indicated to its right, up to the next lower and active elevation.
- **Above elevation:** The color is used for the surfaces whose corresponding elevation is indicated to its right, up to the next higher and active elevation.