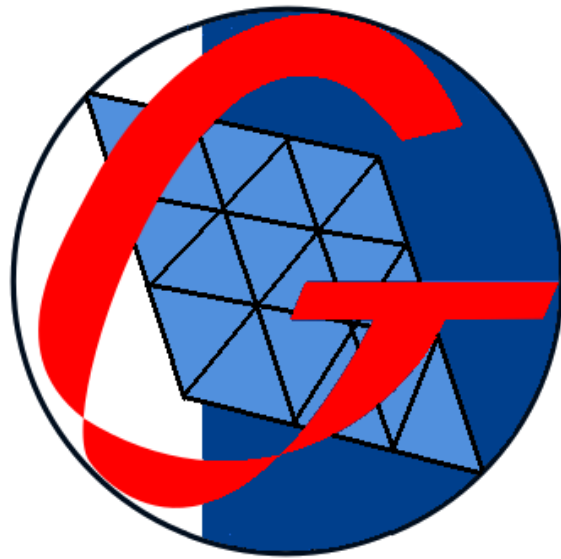


Geo Meshing v4.3



July 2023

Contents

1. Introduction.....	6
2. Main Features	7
2.1. 3D Features	7
2.2. 2D Plane Strain Features.....	8
3. <i>Geo Meshing v4.3</i> Setup	9
3.1. Automatic Installation (recommended)	9
3.2. Manual Installation (not recommended)	10
3.3. Registration.....	10
4. <i>Geo Meshing v4.3</i> Overview	12
4.1. Geo Meshing History	13
5. Main Menu	15
5.1. File Menu	15
5.1.1. Open/Create History	15
5.1.2. Set Number of Processor Menu	15
5.1.3. Output Version	16
5.1.4. Tools	17
5.1.5. Export	21
5.1.6. Import.....	24
5.1.7. Quit.....	25
5.2. FMT (Flat Mesh & Topography).....	26
5.3. Load.....	30
5.4. 3D Surfaces Menu.....	31
5.4.1. 3D Dam Surface	32
5.4.2. 3D Circular Excavation Surface	34
5.4.3. 3D Vertical Fill (MSE Wall) Surface	36
5.4.4. 3D Polygonal Fill Surface (not active with <i>Test License</i>)	38
5.4.5. 3D Polygonal Excavation Surface (not active with <i>Test License</i>).....	40
5.4.6. 3D Finer Mesh Surface (not active with <i>Test License</i>).....	42
5.4.7. 3D Horizontal Finer Mesh Surface (not active with <i>Test License</i>)	45
5.4.8. 3D Interface Surface (not active with <i>Test License</i>)	47

5.4.9.	3D Wedge Excavation Surface (not active with <i>Test License</i>)	49
5.4.10.	3D Add / Subtract Surface (not active with <i>Test License</i>)	51
5.5.	3D Meshing	53
5.6.	Plan3 Strain (not active with <i>Test License</i>)	55
5.7.	Help Menu	57
5.7.1.	Manual Option	57
5.7.2.	Contact Option	57
5.7.3.	About Geo Meshing Option	58
6.	Render	59
6.1.	3D Render Controls.....	59
6.2.	Plane Strain Render Controls.....	60
6.3.	Selecting Coordinates with Mouse	60
7.	FLAC3D® and FLAC® Integration.....	62
8.	Trouble Shooting.....	63
9.	Examples	64
9.1.	Using Geo Meshing for FLAC3D/FLAC2D – Example 1 (Plane Strain models)	64
9.2.	Edit History – Example 1 continuation (Plane Strain Model)	70
9.3.	Using Geo Meshing for FLAC3D – Example 2 (3D models)	75
9.4.	More Examples	82
Figure 1:	<i>Geo Meshing v4.3</i> Installation Dialog Window.	9
Figure 2:	<i>Geo Meshing v4.3</i> Code.....	11
Figure 3:	<i>Geo Meshing v4.3</i> Overview.....	12
Figure 4:	<i>History commands</i>	13
Figure 5:	Geo Meshing History – Run, define range dialog window.....	13
Figure 6:	Geo Meshing History – Edit, edits option in history.	14
Figure 7:	File Menu – Open/Create History.	15
Figure 8:	Open/Create history dialog window.	15
Figure 9:	File Menu – Set Number of Processors.	16
Figure 10:	File Menu – Output Version.	16
Figure 11:	File Menu – Tools.	17
Figure 12:	Stage Capacity Curve dialog window.	18

Figure 13: Interpolate State Parameter dialog window.....	19
Figure 14: State Parameter Statistical Analysis.....	20
Figure 15: Assign Areas dialog window.....	21
Figure 16: File Menu – Export.....	21
Figure 17: <i>Geo Meshing v4.3</i> to CAD script surface dialog window.....	22
Figure 18: <i>Geo Meshing v4.3</i> to Flac3D® Geometry dialog window.....	22
Figure 19: File Menu – Selected Coordinates dialog window.....	23
Figure 20: File Menu – Export.....	24
Figure 21: GID® to <i>Geo Meshing v4.3</i> dialog windows.....	24
Figure 22: File Menu – Quit.....	25
Figure 23: FMT (Flat Mesh & Topography) Menu.....	26
Figure 24: CAD Point Import.....	27
Figure 25: Load Menu.....	30
Figure 26: 3D Surfaces Menu – Dam.....	32
Figure 27: 3D Surfaces Menu – Circular Excavation.....	34
Figure 28: 3D Surfaces Menu – Vertical Fill (MSE Wall).....	36
Figure 29: 3D Surfaces Menu – Polygonal Fill.....	38
Figure 30: 3D Surfaces Menu – Polygonal Excavation.....	40
Figure 31: 3D Surfaces Menu – Finer Mesh.....	42
Figure 32: 3D Surfaces Menu – Horizontal Finer Mesh.....	45
Figure 33: 3D Surfaces Menu – Interface Mesh.....	47
Figure 34: 3D Surfaces Menu – Wedge Excavation.....	49
Figure 35: 3D Surfaces Menu – Add / Subtract.....	51
Figure 36: 3D Meshing Section.....	53
Figure 37: Plane Strain Dialog Window.....	55
Figure 38: 2D Plain Strain Geometry in Render Area.....	56
Figure 39: Help Strip Menu – Manual.....	57
Figure 40: Help Strip Menu – Contact.....	57
Figure 41: Help Strip Menu – About Geo Meshing.....	58
Figure 42: 3D Render Section – Surface Mode.....	59
Figure 43: Dam CAD drawing.....	64
Figure 44: CAD drawing layers.....	64
Figure 45: Geo Meshing – Plane Strain.....	65
Figure 46: History file dialog window.....	65
Figure 47: Plane Strain option – Open.....	66
Figure 48: Open CAD file dialog window.....	67
Figure 49: Plane Strain option – Analyze.....	68
Figure 50: Geo Meshing – 2D wireframe render.....	68

Figure 51: FLAC3D Plane Strain model.	69
Figure 52: History Edit.	70
Figure 53: Open Geo Meshing History File.....	71
Figure 54: Geo Meshing History File Loaded.....	71
Figure 55: Geo Meshing Edit Windows.	72
Figure 56: Layer Height Edited.	72
Figure 57: Run History.	73
Figure 58: Geo Meshing Extended History File.	74
Figure 59: FLAC3D Plane Strain model. Coarser Base.	74
Figure 60: <i>Bathtub</i> sketch.....	75
Figure 61: Notepad initial steps.....	76
Figure 62: FMT option – Geo Meshing v4.3.	77
Figure 63: “F” view in render area – Geo Meshing v4.3.	77
Figure 64: Polygonal excavation – Geo Meshing v4.3.....	78
Figure 65: Geo Meshing – <i>bathtub</i> surface.	78
Figure 66: 3D Meshing Window – base meshing.	79
Figure 67: <i>Bathtub</i> base mesh – image from FLAC3D©.....	80
Figure 68: 3D Meshing Window – fill meshing.....	80
Figure 69: <i>Bathtub</i> full mesh. Upper image, full mesh in slot 1. Lower image, full mesh in slot 2 – image from FLAC3D©.	81

1. Introduction

Geo Meshing v4.3 is a software specifically designed for developing 3D meshes for FLAC3D® and 2D meshes for FLAC®.

Geo Meshing v4.3 greatly simplifies the complex process of producing advanced 3D meshes for FLAC3D®, and therefore, reduces the cost associated with those tasks. Likewise, *Geo Meshing v4.3* effortlessly produce meshes for FLAC®, automatically computing grid elements number, ratios and geometries.

Although *Geo Meshing v4.3* was specifically tailored for FLAC3D®/ FLAC®, it can also be used with other software by post processing and modifying the output files.

2. Main Features

2.1. 3D Features

- **Layered mesh:** *Geo Meshing v4.3* can automatically produce horizontal layers as it meshes bodies. In addition, it can name each layer with sequential designations, thus construction sequences can be simulated.
- **Non-Chaotic mesh generation:** *Geo Meshing v4.3* creates, as much as possible, balanced meshes, where large contrasts in element sizes and orientations are avoided.
- **Parametric body definition:** *Geo Meshing v4.3* uses a parametric definition for creating a wide variety of bodies without the need of a CAD assistant software.
- **CAD compatibility:** *Geo Meshing v4.3* can extract polyline information from DXF files. Using this feature, a topographic landform can be developed in minutes.
- **Bing Maps® Ready:** *Geo Meshing v4.3* can extract topographic surfaces directly from Bing Maps®, thus landforms can be easily imported into *Geo Meshing v4.3*.
- **GID® compatibility:** *Geo Meshing v4.3* reads and imports 2D files created with GID®. These files can be used to produce 3D meshes.
- **Meshing refinement:** *Geo Meshing v4.3* can refine meshes at any level within the model.
- **Interpolation algorithms:** *Geo Meshing v4.3* includes two interpolation algorithms, inverse distance squared and Kriging for obtaining smooth surface profiles.
- **Flexible:** *Geo Meshing v4.3* output files can be post processed and used with other software.
- **Multiprocessor:** *Geo Meshing v4.3* is a parallel processing software, allowing for faster development of meshes.
- **Export to FLAC3D® Geometry:** Any surface developed with *Geo Meshing v4.3* can be transformed and imported into FLAC3D® as a 3D geometry.

- **Compatible:** *Geo Meshing v4.3* is capable of producing meshes compatible with current and previous FLAC3D® versions.

2.2. 2D Plane Strain Features

- **Layered mesh:** *Geo Meshing v4.3* can automatically produce horizontal layers as it meshes bodies. In addition, it can name each layer with sequential designations, thus construction sequences can be simulated.
- **Non-Chaotic mesh generation:** *Geo Meshing v4.3* creates, as much as possible, balanced meshes, where large contrasts in element sizes and orientations are avoided.
- **CAD compatibility:** *Geo Meshing v4.3* can extract polyline information from DXF files. Using this feature, elements sizes and model domain can be determined.
- **Easy to edit:** *Geo Meshing v4.3* generates a *.gmh file that can be edited in *Geo Meshing v4.3*. Using the history file allows to further process the mesh and tailored it to a specific analysis.
- **Multiprocessor:** *Geo Meshing v4.3* is a parallel processing software, allowing for faster development of meshes.
- **Export to FLAC3D® Geometry:** Any surface developed with *Geo Meshing v4.3* can be transformed and imported into FLAC3D® as a 3D geometry.

3. Geo Meshing v4.3 Setup

Geo Meshing v4.3 can be installed automatically or manually. It is highly recommended to install it automatically.

Once *Geo Meshing v4.3* is installed in your computer, the files described in section 3.2 will show at the installation folder. In addition, a file called “Geo_meshing.key” also needs to be stored in this folder. See section 3.3 for instructions.

3.1. Automatic Installation (recommended)

Once you have downloaded the installation file, unzip the file and double click on it. You will see a welcome window as the one in [Figure 1](#). Follow the screen instructions to install *Geo Meshing v4.3*.

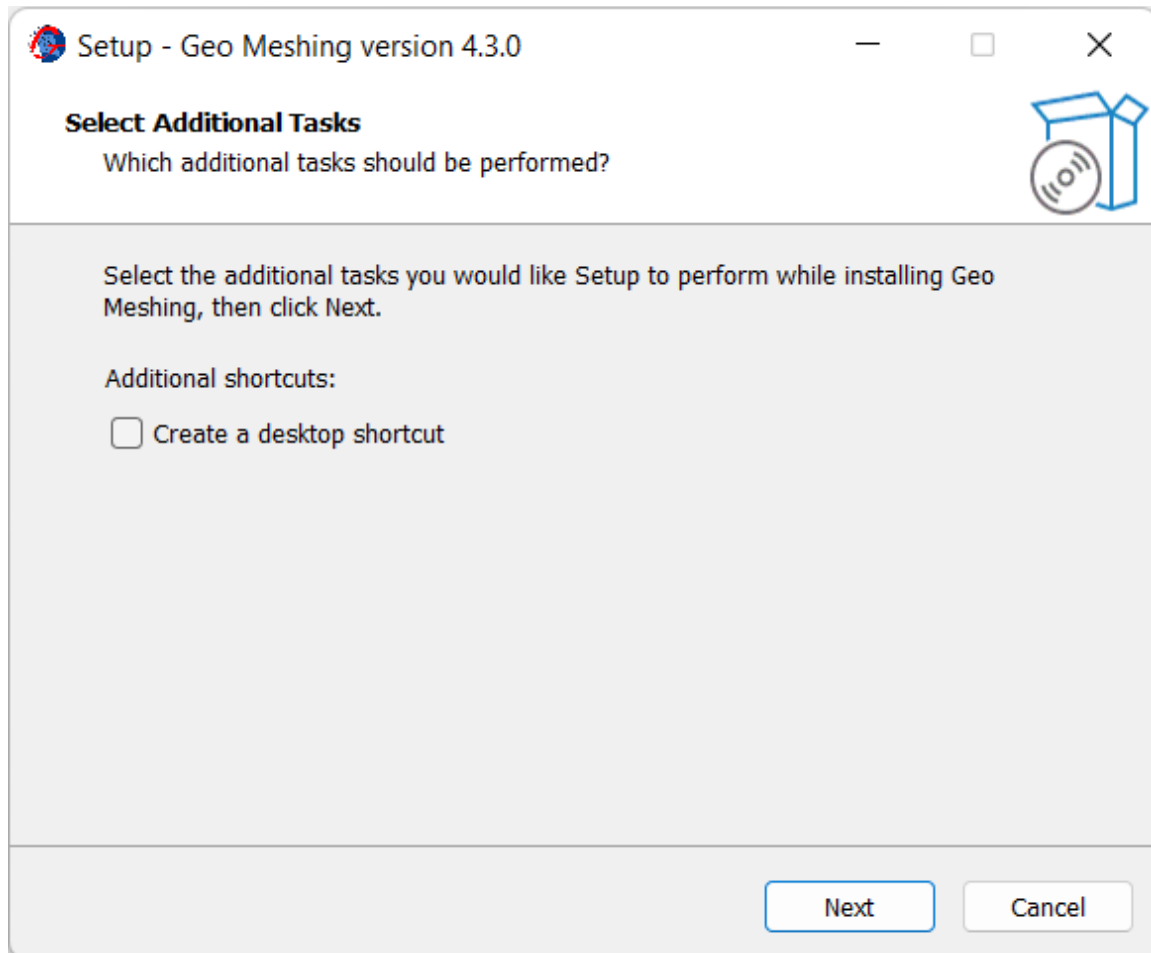


Figure 1: *Geo Meshing v4.3* Installation Dialog Window.

3.2. Manual Installation (not recommended)

Create a folder in C: \ Program Files (x86) and name it “Geo Meshing”. Copy and unzip the files to the directory:

C:\Program Files (x86)\Geo Meshing

Warning: Please notice that [Geo Meshing v4.3](#) will not run properly if copied to a different source folder.

This folder should contain the following files:

- *Geo Meshing v4.3.x.exe*
- *GM_43x.exe*
- *topo_render_v5.dll*
- *2D_render.dll*
- *CPTu_plot_v2.dll*
- *bmaplib.dll*
- *DXF_Sketcker.dll*
- *Microsoft.Maps.MapControl.WPF.dll*
- *log_change.txt*
- *GMMannual4.3.pdf*
- *Resources [folder]*
- *MapData [folder]*
- *Examples [folder]*

3.3. Registration

Geo Meshing v4.3 needs the file “Geo_meshing.key” to run. Please send an email to geomeshing@gmail.com to either request a trial license for 15 days or to purchase an annual license.

You will also need to include in your email the *Geo Meshing v4.3* code. The *Geo Meshing v4.3* code can be obtained under *Help/About Geo Meshing* menu, AFTER the program has been installed (see section 3).

To avoid mistyped information, highlight *Geo Meshing v4.3* code and then right click on it, copy/paste into your email. This file needs to be stored in *C:\Program Files (x86)\Geo Meshing*. After you have copied the license file in the software folder, you will need to restart *Geo Meshing v4.3* to active it.

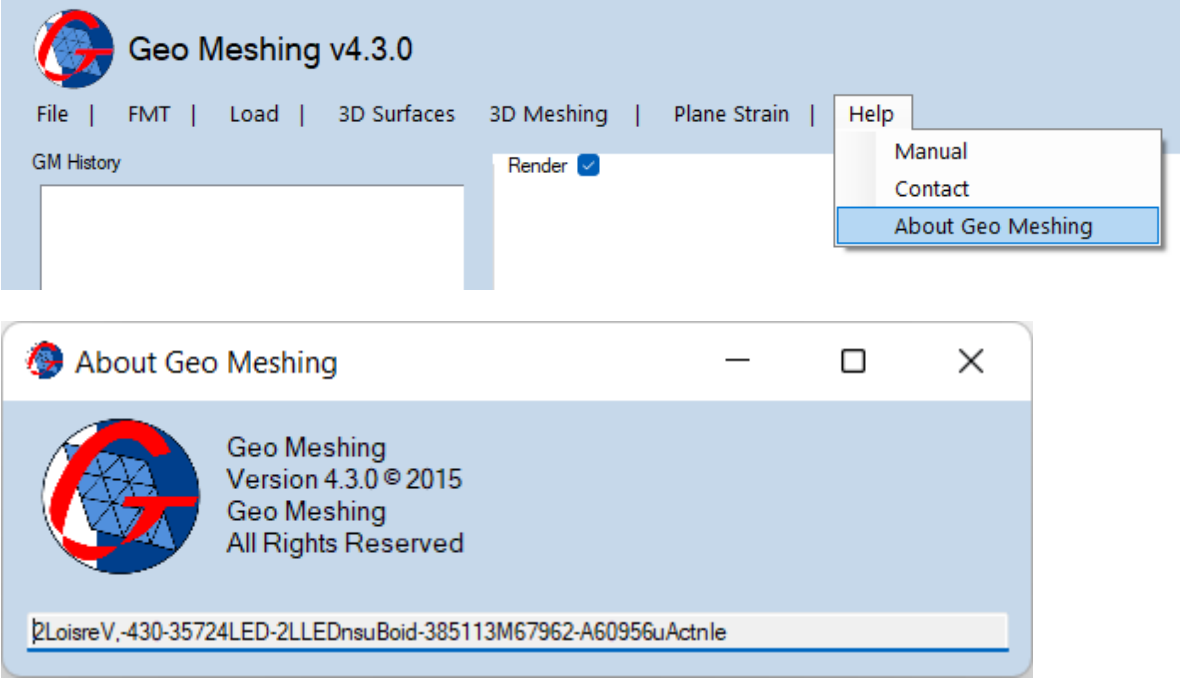


Figure 2: *Geo Meshing v4.3* Code.

4. Geo Meshing v4.3 Overview

Figure 3 shows the main components of *Geo Meshing v4.3*.

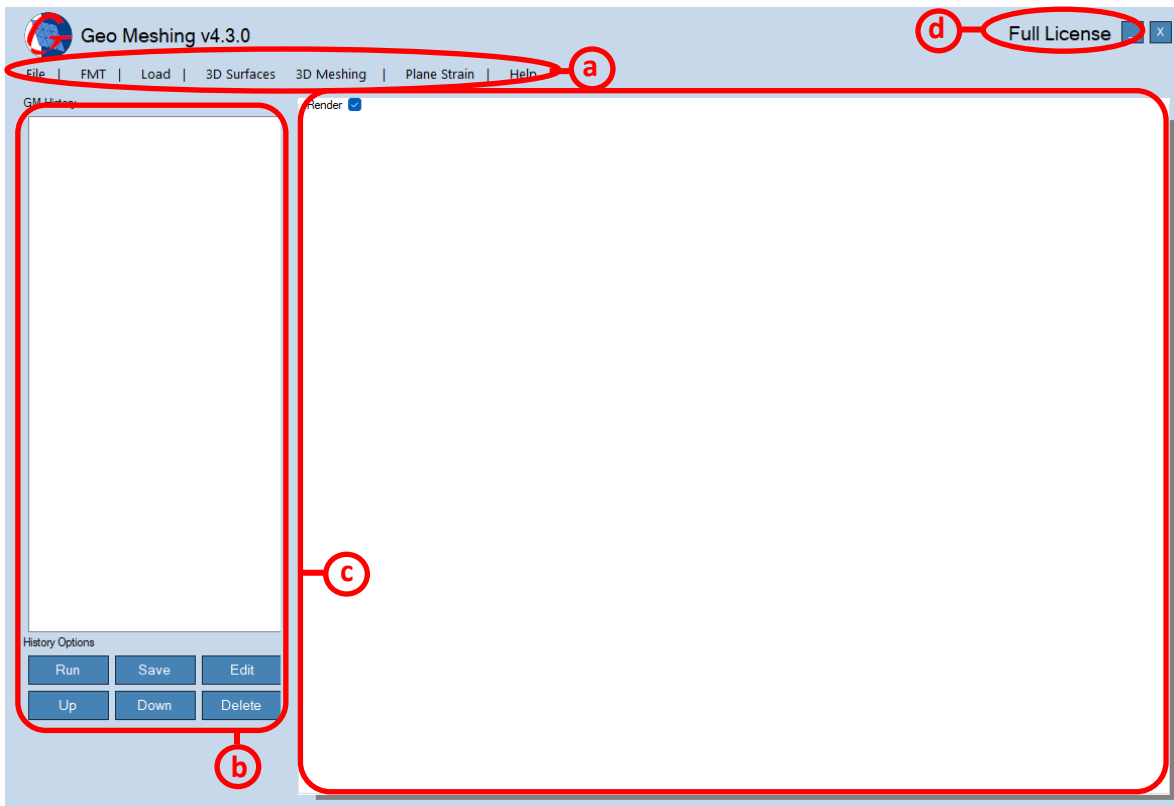


Figure 3: *Geo Meshing v4.3* Overview.

- a) **Main Menu:** The strip menu area provides options for *Geo Meshing v4.3*. Here one can configure, create and manage meshes. In addition, one can find the manual, the contact information and license code under the help menu (see corresponding sections).
- b) **GM History:** This area tracks history of the meshing events (see section 4.1 and 5.1.1).
- c) **Render:** This window displays information for 2D and 3D options. When using 3D options, the last surface/mesh generated with *Geo Meshing v4.3* is shown. This procedure is called after the *3D Meshing* algorithm is executed (see section 5.5). When using the 2D options, this window displays the 2D imported geometry. Finally, *Geo Meshing v4.3* can show surface/mesh or 2D geometry when *Load* option is used (see section 5.3)
- d) **License type:** This area displays the type of license under use in *Geo Meshing v4.3*, it can be “No License”, “Test License” or “Full License” (as shown).

Note: Test license is often referred as Free Version

4.1. Geo Meshing History

- Geo Meshing history has eight options, which are shown in [Figure 4](#).



Figure 4: *History commands.*

- **Run:** This option runs a series of events defined by a range. [Figure 5](#) shows the dialog windows for defining the range. *From* and *To* are the numbers shown in the [Geo Meshing v4.3](#) history windows. Before using this option, select an event in the history window.

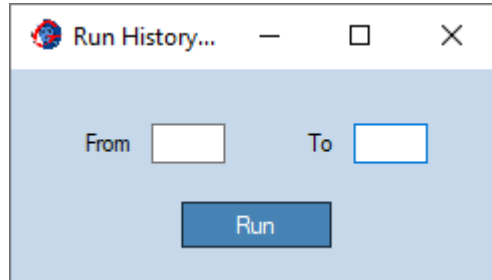


Figure 5: *Geo Meshing History – Run, define range dialog window.*

- **Save:** This option saves the entire [Geo Meshing v4.3](#) history to the hard drive. If a file name for the [Geo Meshing v4.3](#) history has not been defined, one will be required during the saving process.
- **Edit:** This option enables to edit the recorded events in the history window (section 4b)). [Figure 6](#) shows the dialog windows for this option. Currently, only “3D Meshing” events are editable.

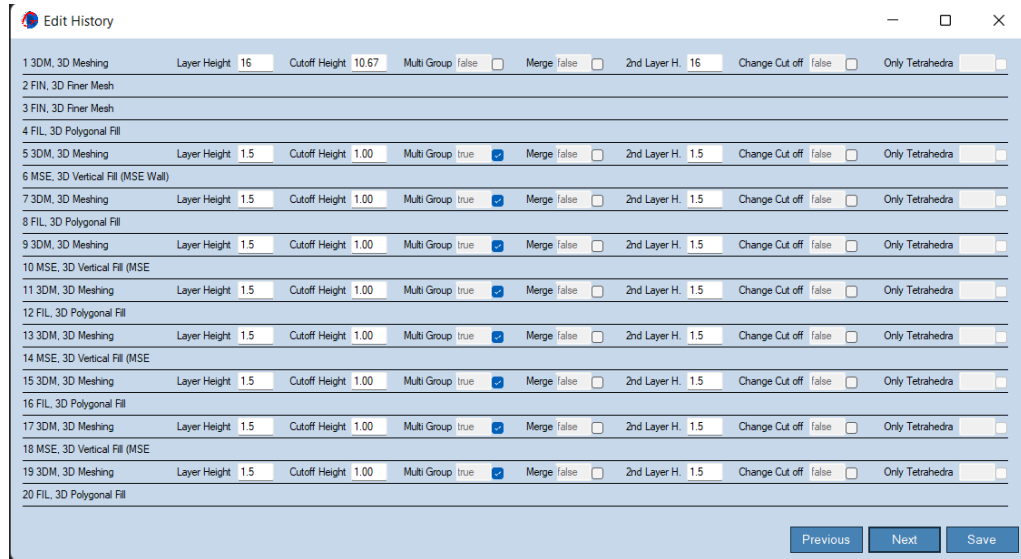


Figure 6: Geo Meshing History – Edit, edits option in history.

- **Up:** This option moves up the current selected event in the history window. It swaps the event position with the above one. Every time the button is pressed, it moves up the current event one position.
- **Down:** This option moves down the current selected event in the history window. It swaps the event position with the below one. Every time the button is pressed, it moves down the current event one position.
- **Delete:** This option deletes the event selected in the *Geo Meshing v4.3* history window. Before using this option, select an event in the history window.

5. Main Menu

5.1. File Menu

5.1.1. Open/Create History

- **File Menu, Open/Create history:** This option (shown in [Figure 7](#)) shows the dialog window of [Figure 8](#), for opening a history file or setting up a new history file.

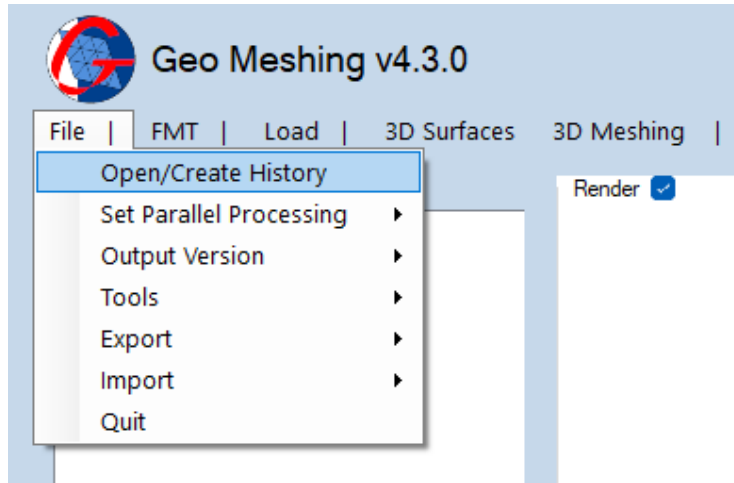


Figure 7: File Menu – Open/Create History.

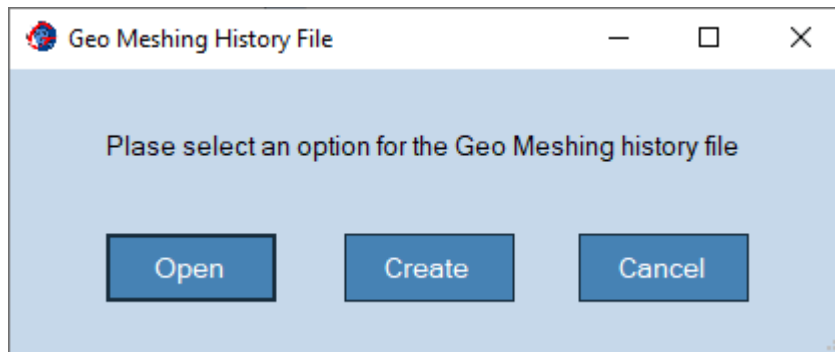


Figure 8: Open/Create history dialog window.

5.1.2. Set Number of Processor Menu

- **File Menu, Set Parallel Processing:** This option (shown in [Figure 9](#)) has a submenu with two options.

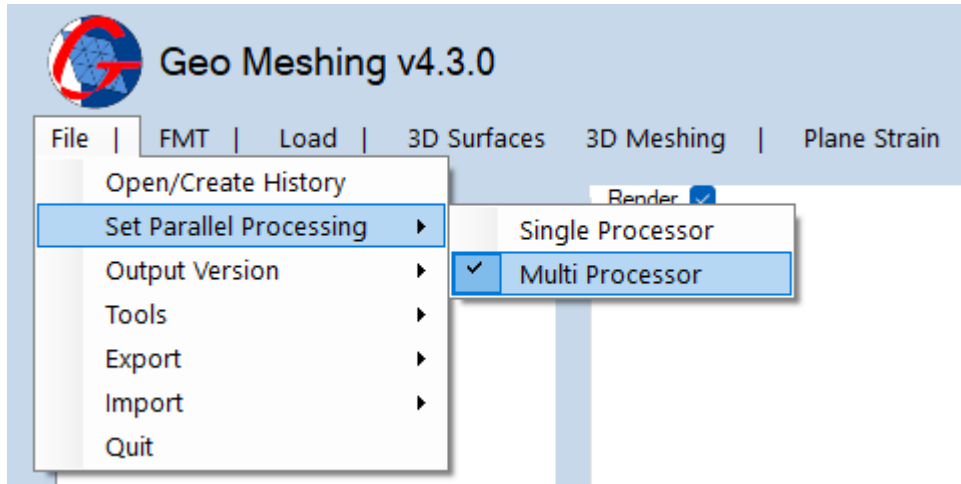


Figure 9: File Menu – Set Number of Processors.

- **Single Processor:** Select this option to use one processor to develop meshes.
- **Multi Processors (default):** Select this option to use parallel processing to develop meshes (not active with Test License). *Geo Meshing v4.3* uses the maximum number of threads available.
 - **Hint:** The larger the number of processors the faster the meshing process. Make sure this setting is set less than or equal to the maximum number of thread in your PC.
 - **WARNING:** Proper ventilation must be provided to the PC/WS/Notebook/Laptop, otherwise it may overheat.

5.1.3. Output Version

- **File Menu, Output Version:** This option (shown in Figure 10) has a submenu with two options for defining the output format.

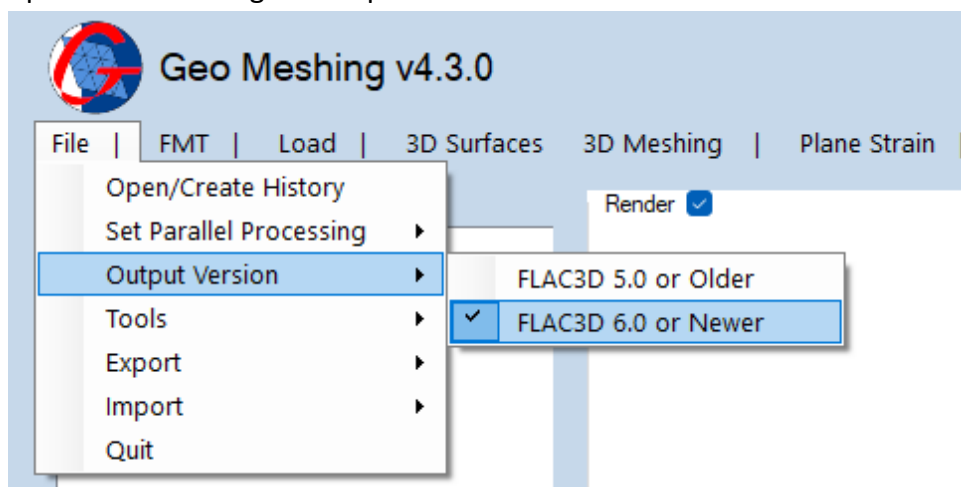


Figure 10: File Menu – Output Version.

- **Flac3D 5.0 or Older:** Select this option to obtain an output file compatible with Flac3D© 5.0 or older format.
- **Flac3D 6.0 or Newer:** Select this option to obtain an output file compatible with Flac3D© 6.0 or newer format.

5.1.4. Tools

- **File Menu, Tool:** This option (shown in [Figure 11](#)) has a submenu with three options.

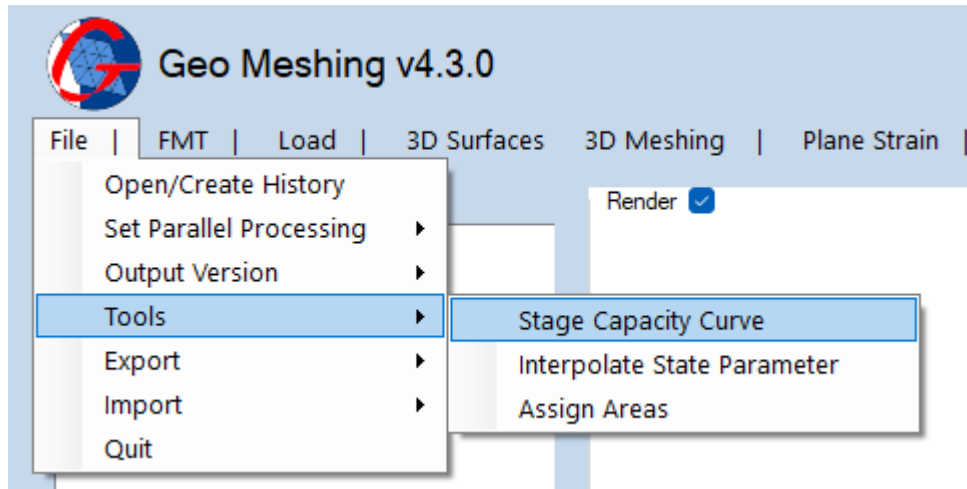


Figure 11: File Menu – Tools.

- **Stage Capacity Curve:** Opens the dialog window shown in [Figure 12](#). This feature computes the volume versus elevation between two 3D surfaces (section 5.4) and records the data in a text file.
 - **Open Top Surface:** Opens a *Geo Meshing v4.3* file, with the 3D surface containing the higher elevation.
 - **Output Bot. Surface:** Opens a *Geo Meshing v4.3* file, with the 3D surface containing the lowest elevation.
 - **Curve Increment:** Set the interval for the points in the stage capacity curve. This will be modified internally to fit the elements size of the meshes.
 - **Generate:** Starts the process.

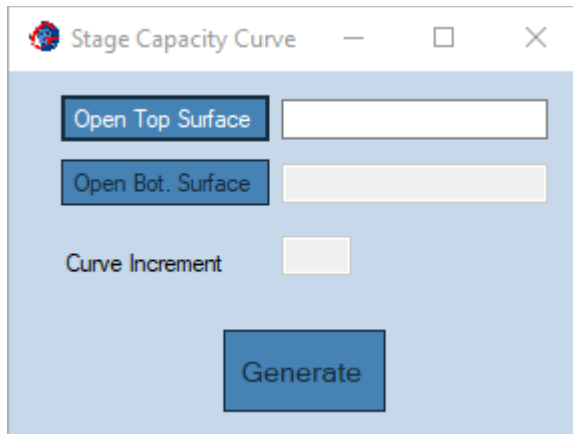


Figure 12: Stage Capacity Curve dialog window.

- **Interpolate State Parameter:** Opens the dialog window shown in [Figure 13](#). This option interpolates the state parameter for an already existing FLAC3D® model. This option can use a state parameter (from example, for a pre processed Excel spreadsheet) and interpolate the state parameter across the mesh domain. Also, it can use advance options, such as either Jefferies (1992) and Plewes (1992), or Shuttle and Cunnings (Widget, 2007) and internally compute the state parameter. Using these last two options over more traditional methods, allows taking into consideration the variability of the stress ratio or K0 conditions.
 - **Open Zone Data:** Opens a file with Zones 3D information. Review the onscreen help icon for more information.
 - **Open DataBase:** Opens a file with CPTU information for computing the state parameter.
 - **Mtc:** Is the critical state slope.
 - **Min Value, Max Value:** These values define where the interpolation is truncated.

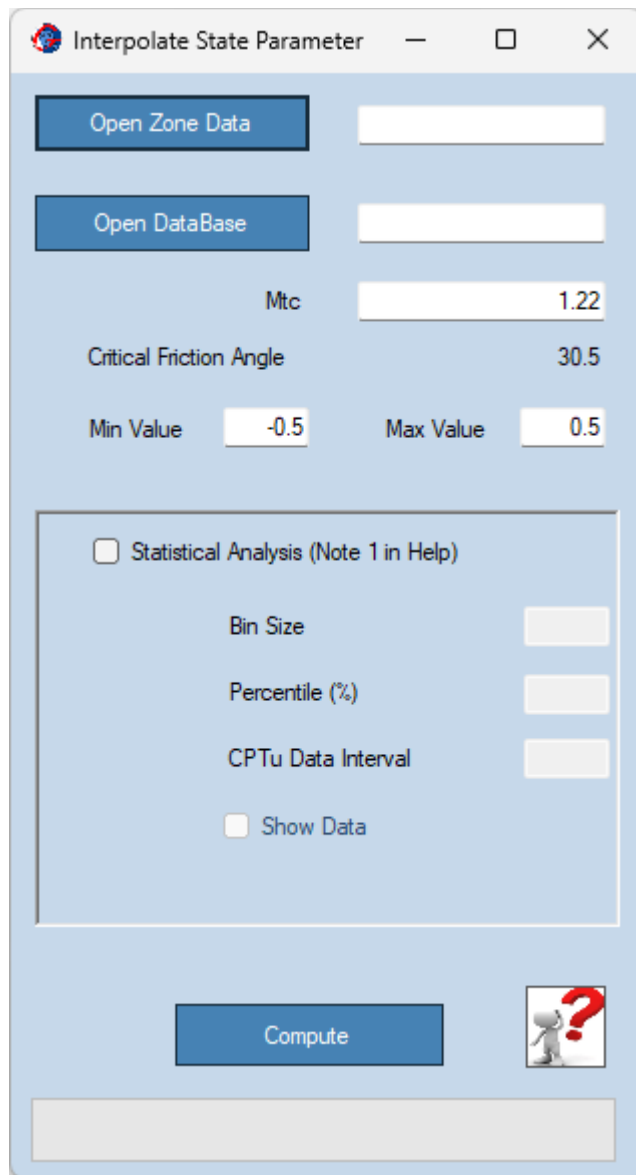


Figure 13: Interpolate State Parameter dialog window.

- **Statistical Analysis:** This option allows statistical analysis over the CPTU state parameter. Instead of using a direct input from the CPTU, bin and percentile analysis are included for statistically pre processing the state parameter before interpolation throughout the Flac3D© mesh.
 - Bin Size:** Range size to consider for the statistical analysis.
 - Percentile:** defines a value larger than a given percentage of the sample. For example, a percentile 70% means that 70% of the values in the bin are less than the selected value.
 - CPTu Data Interval:** Sample acquisition interval of the CPTu.

- **Show Data:** Shows the CPTu data and statistical analysis as shown in Figure 14 (only available when Statistical Analysis is selected).
- **Help Icon:** Provides more detailed information for this option.
- **Compute:** Starts the process.

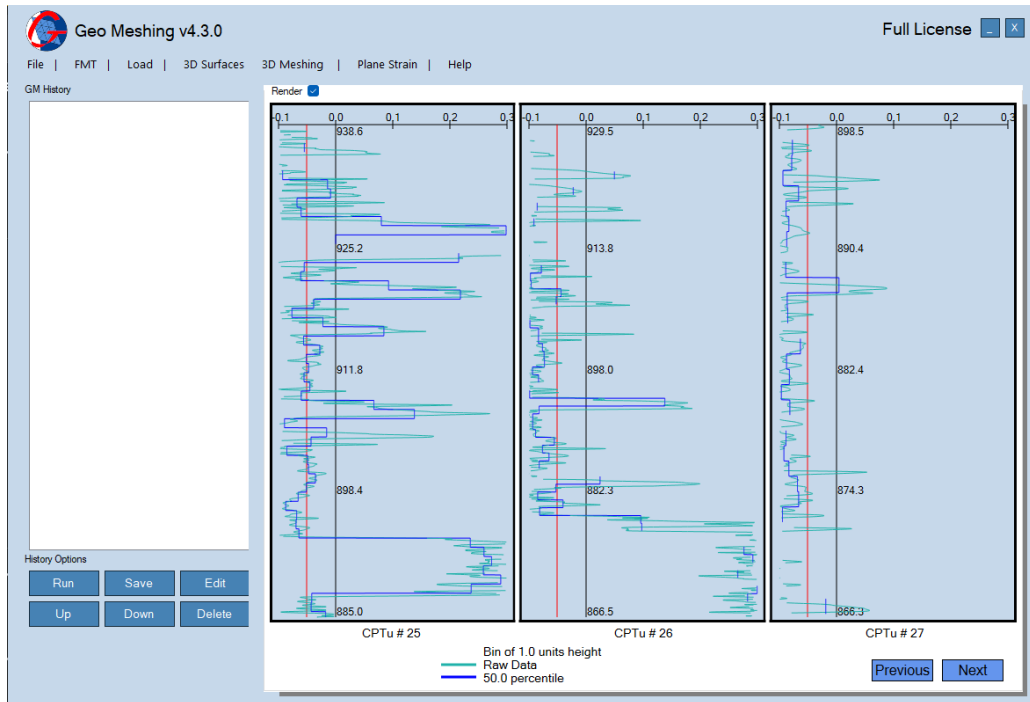


Figure 14: State Parameter Statistical Analysis.

- **Assign Areas:** Opens the dialog window shown in Figure 15. This option Assign Areas to an already existing FLAC3D® mesh. Currently, only for 2D Meshes.
 - **Open Zone Data:** Opens a file with Zones 3D information. Review the onscreen help icon for more information.
 - **Open Polylines:** Opens a file containing the N polylines for defining the areas.
 - **Help Icon:** Provides more detailed information for this option.
 - **Compute:** Starts the process.

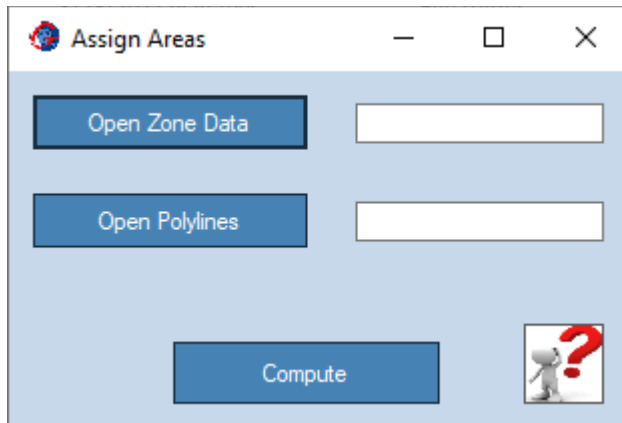


Figure 15: Assign Areas dialog window.

5.1.5. Export

- **File Menu, Export:** This option (shown in Figure 16) has a submenu with three options; *GM to CAD Script*, *GM to FLAC3D® Geometry and Selected Coordinates*.

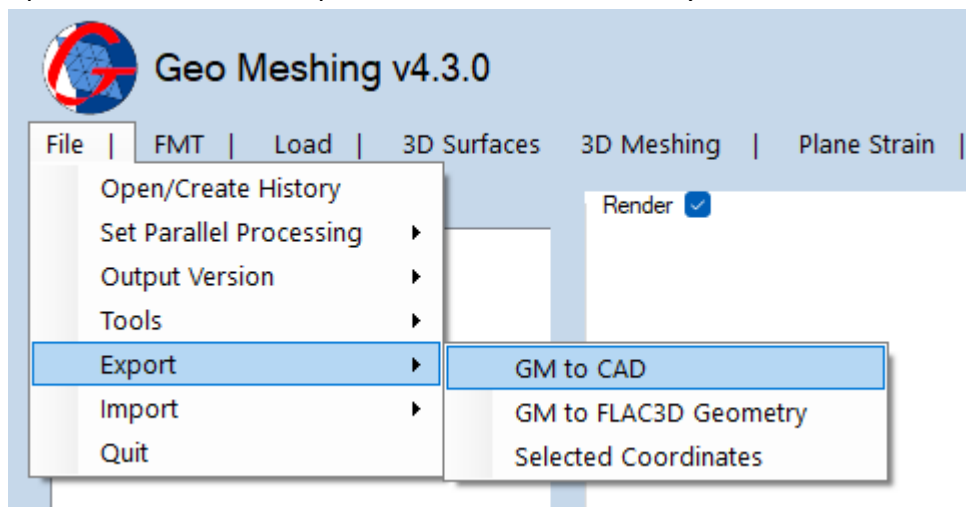


Figure 16: File Menu – Export.

- **GM to CAD Script:** Opens a dialog window (shown in Figure 17) for converting *Geo Meshing v4.3* meshes to a CAD script surface. The script can be drag/dropped into CAD software and it will create a 3Dface surface.
 - **Open GMM File:** Opens a *Geo Meshing v4.3* file.
 - **Output File Name:** Sets the name for the script file.
 - **Convert:** Starts the process.
 - **Hint:** To avoid miss location of the cursor during drawing 3Dfaces, deactivate object snap in your CAD software (usually by pressing F3 key).

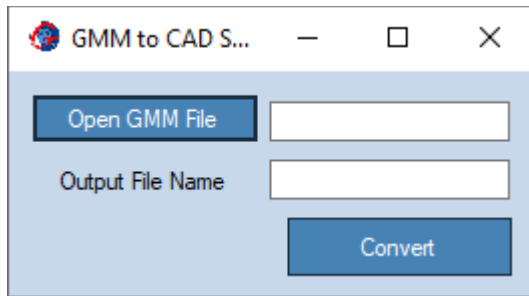


Figure 17: *Geo Meshing v4.3* to CAD script surface dialog window.

- **GM to Flac3D® Geometry:** Opens a dialog window (shown in Figure 18) for converting *Geo Meshing v4.3* meshes to Flac3d® geometry. The resulting file can be imported in Flac3D® using *Open Item/Geometric Data: Import* option.
 - **Open GMM File:** Opens a *Geo Meshing v4.3* file.
 - **Output File Name:** Sets the name for the Flac3D® geometry file.
 - **Group Name:** Sets the group name for Flac3D®.
 - **Slot Number:** Sets the slot number to be used by *Group Name*.
 - **Convert:** Starts the process.

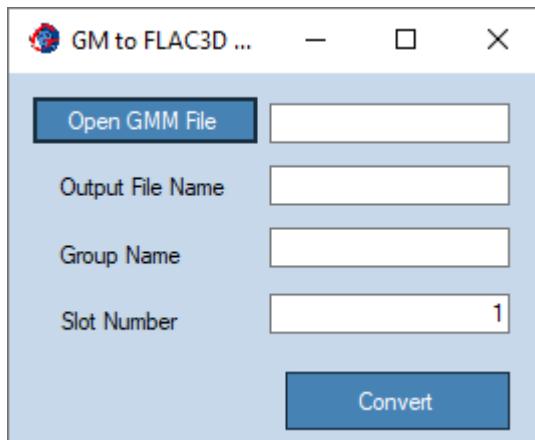


Figure 18: *Geo Meshing v4.3* to Flac3D® Geometry dialog window.

- **Selected Coordinates:** This option shows the dialog window of Figure 19, for creating a file with selected XYZ coordinates. Coordinates are selected using the render area (Section 6.2).
 - **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the coordinates are obtained.
 - **Suggested Output File Name:** As indicated.
 - **Xi, Yi and Zi Coordinates:** These coordinates are displayed as one clicks on the render area.

- **Hint:** xyz coordinates can be selected using the mouse and the 3D render “F” projection option (see section 6.2).
- **Reset:** Deletes the current coordinates.
- **Generate:** Export xyz coordinates.

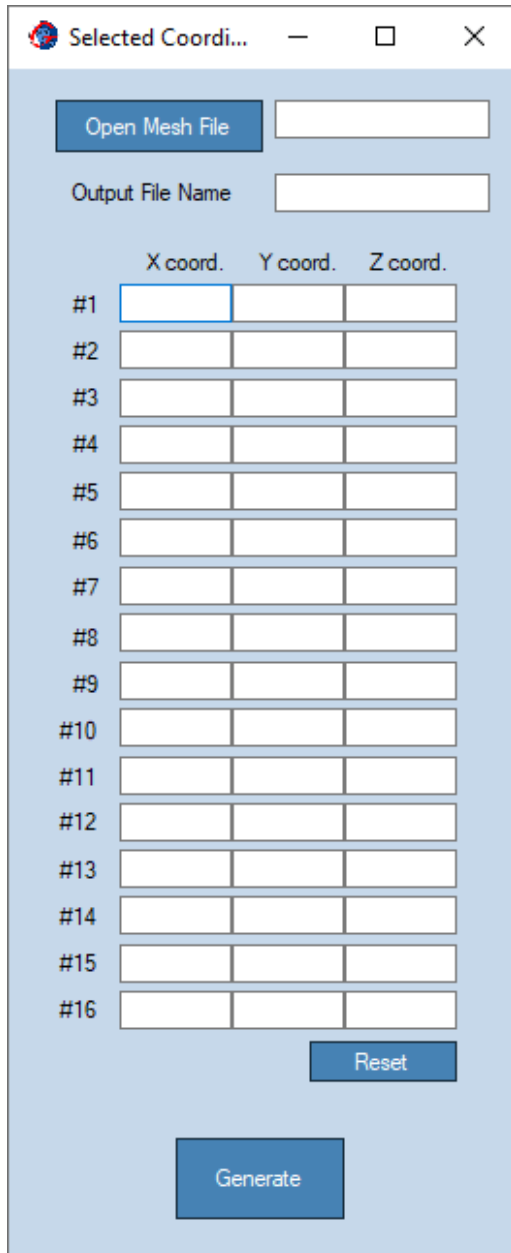


Figure 19: File Menu – Selected Coordinates dialog window.

5.1.6. Import

- **File Menu, Import:** This option (shown in [Figure 20](#)) has a submenu with *GID to GM* option.

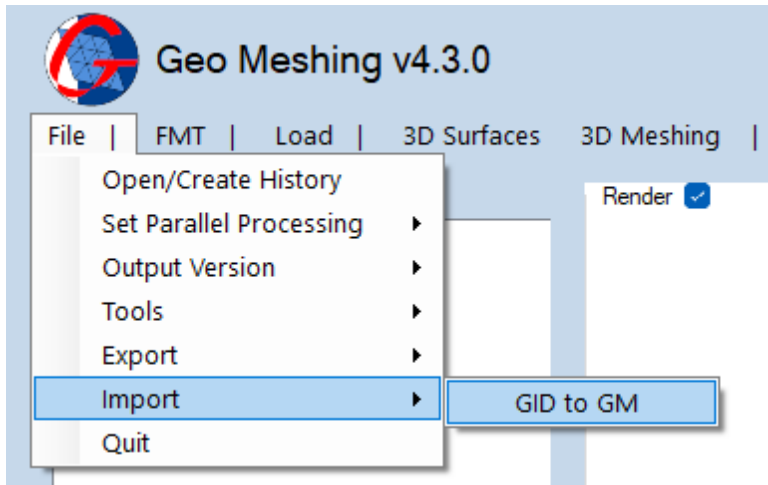


Figure 20: File Menu – Export.

- **GID® to GM:** Opens a dialog window (shown in [Figure 21](#)) for converting GID® meshes files (*.msh) to *Geo Meshing v4.3* files (*.gmm). This program only works with 2D meshes exported from GID®. More information can be obtained at www.gidhome.com/.
 - **Open GID File:** Opens a GID® file.
 - **Output File Name:** Sets the name for the GM mesh file (*Geo Meshing v4.3* Mesh).
 - **Convert:** Starts the process.

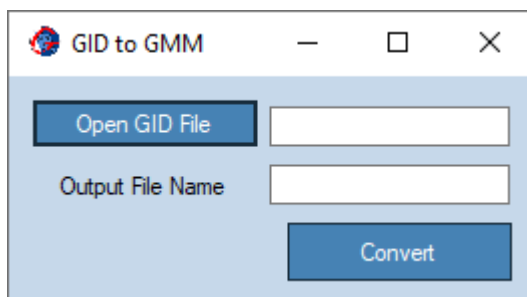


Figure 21: GID® to *Geo Meshing v4.3* dialog windows.

5.1.7. Quit

- **File Menu, Quit:** Exits from *Geo Meshing v4.3* (shown in [Figure 22](#)).

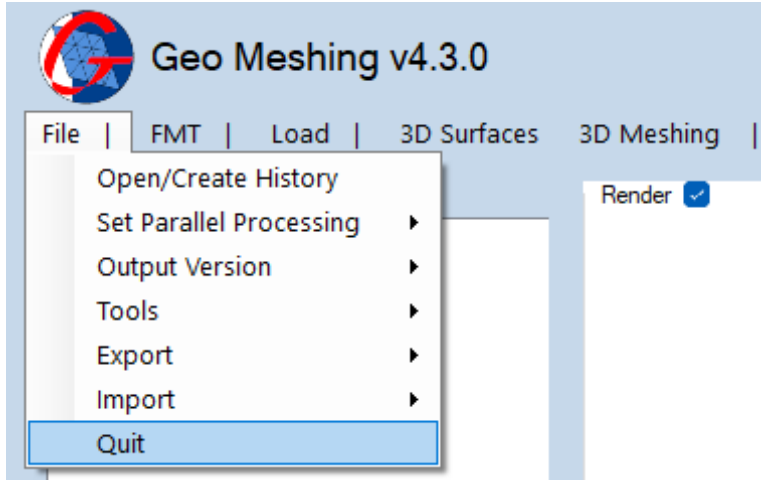


Figure 22: File Menu – Quit.

5.2. FMT (Flat Mesh & Topography)

This option opens the dialog window shown in Figure 23.

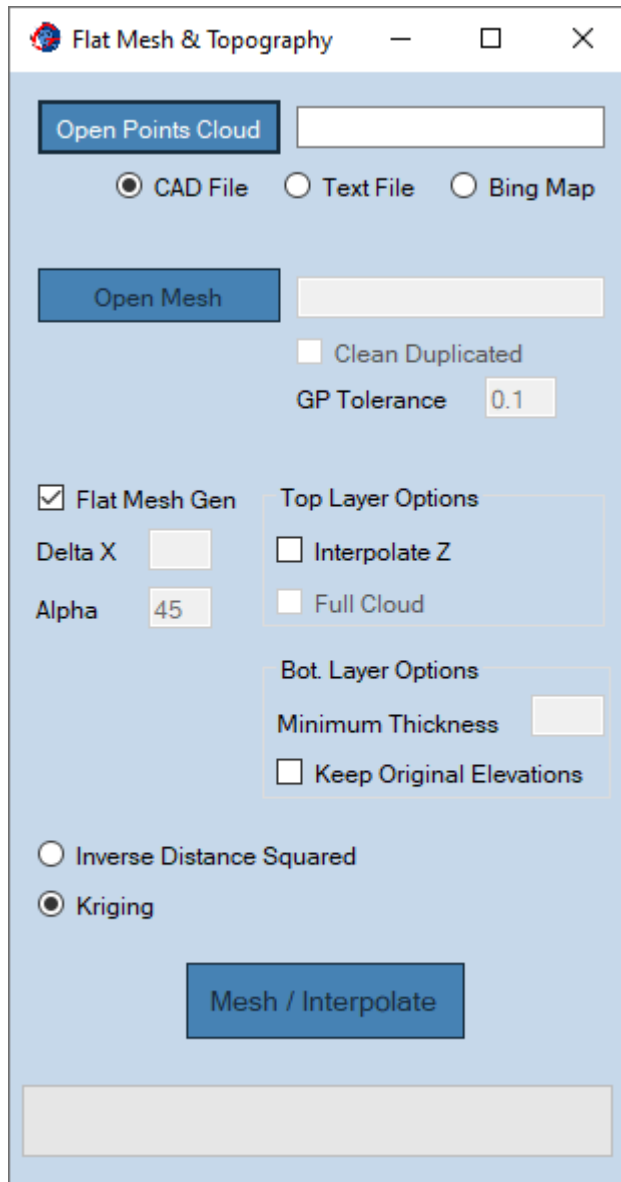


Figure 23: FMT (Flat Mesh & Topography) Menu.

- **Open Points Cloud:** This option helps selecting a cloud of point that represents a surface. Using this cloud of points a *Geo Meshing v4.3* mesh can be interpolated.

There are three ways of selecting/creating the cloud of points; by selecting a CAD file, by selecting a *.txt file and by creating the cloud of points from Bing Map®.

- **CAD File:** If this option is selected the dialog window shown in [Figure 24](#) is displayed. The cloud of points will be created from a set of 2D/3D polylines from a specific layer.
 - **Open CAD File:** Select a DXF file for gathering surface information. *Geo Meshing v4.3* will extract vertex information from all layers.
 - **Warning:** The CAD file (*.DXF) MUST be saved in R12 version, or data won't be read.
 - **Select Layer:** Select the layer from which vertex are going to be imported into *Geo Meshing v4.3*, use the dropdown menu.
 - **Point Spacing:** Input a real value for vertex spacing, S. If the spacing between vertex in the polyline is larger than S, a linear interpolation is performed for intermediate points.
 - **Generate Data Point:** Starts data/cloud of points generation.

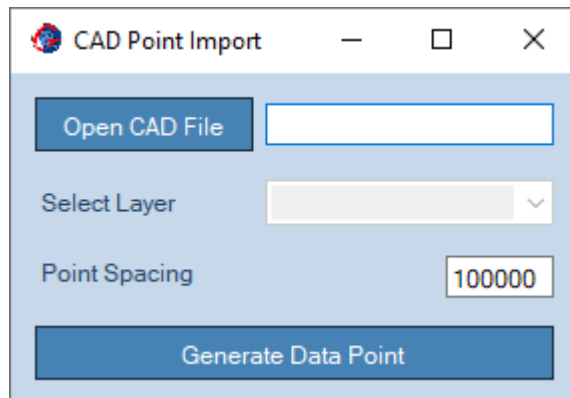


Figure 24: CAD Point Import.

- **Text File:** If this option is selected, opens a dialog window for importing a *.txt file containing the cloud of points representing a surface.

Notice: the cloud of point format is:

```
X1 Y1 Z1
X2 Y2 Z2
X3 Y3 Z3 ...
```
- **Bing Maps:** Opens a dialog window for saving Bing Maps topographic information into a *.txt file. The file, after being processed by Bing Maps®'s services, will contain the cloud of points representing a surface (not active with *Test License*).

Be aware that maps generated with this option are low resolution maps and might be outdated.

Important!! In order of use this service, a Bing Maps key need to be obtained. Please visit <https://www.bingmapsportal.com/> for more details.

(Once the key is obtained, please copy your key number to the text file located at C:\Program Files (x86)\Geo Meshing\MapData\Bing_Key.txt)

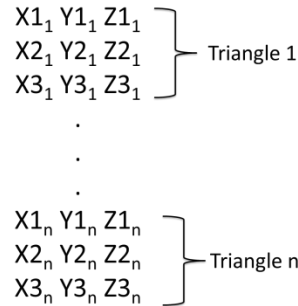
Important!! Do not share your key with others; it is for your personal use only.

- **Open Mesh:** See Flat Mesh Gen option.
 - **Clean Duplicated:** If this option is selected, *Geo Meshing v4.3* will check the entire mesh and it will merge any two grid points at a distance equal or less than *GP Tolerance* (See Flat Mesh Gen).
 - **GP Tolerance:** See above.

- **Flat Mesh Gen:** When this option is checked, a simple homogeneous flat mesh will be generated based on the input data. The extension of the flat mesh is automatically computed from the cloud of point information. The size and shape of the elements is determined by *Delta X* and *Alpha*. *Delta X* determines the side size of the triangle unit, while *Alpha* determines one of the three internal angles. By default, one of the angles is 90°, the other two angles are defined by *Alpha* and the difference of 90° – *Alpha*. *Delta X* should be in the same units as the cloud of point data (meters, feet, etc.) and *Alpha* is in degrees. *Alpha* equal to 45° creates a homogeneous flat mesh with isosceles rectangular triangles. If this option is not checked, the user must provide a flat mesh arrangement of triangles.

Unchecking this option enables *Open Mesh* and *GP Tolerance* options. The first option allows for selecting a custom flat mesh (*.gmm file), while the second option sets the tolerance at which nearby GP points are merged. This last option is useful when “hand modified” flat meshes have been created. It is recommended that the user use 1/20 of your smallest element side for *GP Tolerance*.

The custom mesh should have the following format:



Where $Xi_n \ Yi_n \ Zi_n, i=1,2,3$ represent the triangle three corners. The mesh is composed of n triangles.

- **Top Layer Options:**

- **Interpolate Z:** When this option is checked, the elevation for each corner of the triangles in the flat/custom mesh, is interpolated using the cloud of points. This option will produce two data files, one with extension "..._bot.gmm" and the other with extension "..._top.gmm". These files represent two surfaces, the bottom (see below, *Bot. Layer Options*) and top surfaces, respectively. The top surface corresponds to the formal flat mesh (no longer flat), interpolated using the cloud of points data.

Important!! The grid order is altered after each use of the *Interpolate Z* option. Therefore, you must use your_archive_bot.gmm for *Open Bot.* option in *3D Meshing* tool, when generating more meshes.

- **Full Cloud:** This option sends the entire cloud of points to each processor (when multiple processors are selected, 5.1.2). This option is useful when the topography is relative flat or has few points and interpolation problems have arisen.

- **Bot. Layer Options:**

- **Minimum Thickness:** This value defines the bottom layer constant elevation for the bottom mesh ("..._bot.gmm"). The bottom surface can be a flat mesh arrangement with constant elevation and equal to the difference of $[Minz - Minimum\ Thickness]$. *Minz* is automatically computed from the cloud of points data, and corresponds to the minimum elevation of the data set. This option is useful for setting the model bottom elevation. Alternatively, the bottom surface can be left unaltered, preserving the original elevation (see below).

- **Keep Original Elevations:** Check this box if original surface elevations are meant to be preserved. This is useful when a new interpolated mesh is placed over an irregular mesh, for example, a topography.

Two algorithms are provided for interpolating points, *Inverse Distance Square* and *Kriging*. The later produces smoother results, but the former is better when the cloud of points has few points.

- **Mesh/Interpolate:** Starts the Surface Mesh process.

5.3. Load

This option opens the dialog window shown in [Figure 25](#).

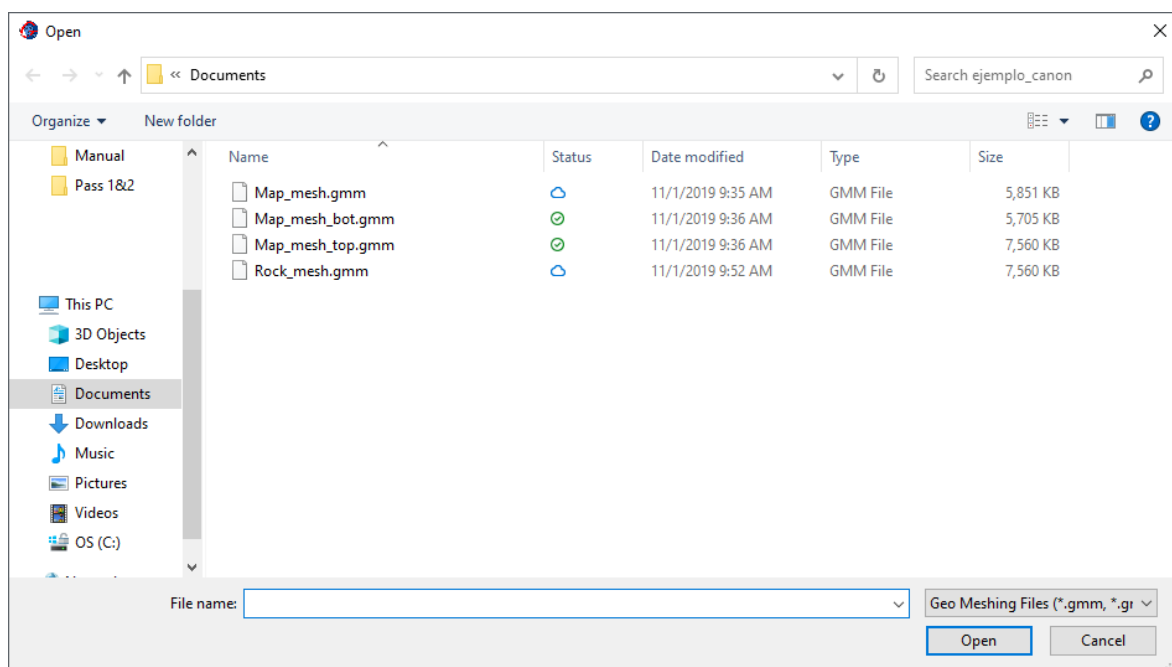


Figure 25: Load Menu.

- Navigate to the folder where you have your *Geo Meshing v4.3* files stored.
- Select either *.gmm file (for 3D surfaces) or *.gm2d file (for 2d wireframes).
- The selected surface or geometry is shown in the *render* area.

5.4. 3D Surfaces Menu

This option creates a landform over any given surface. All landforms are defined parametrically, thus there is no need to develop complex CAD models to obtain a 100% compatible landform into the topography. Once the landform is generated, the *3D Meshing* (section 0) needs to be executed to create the mesh (except for the *Finer Mesh* and *Interface* option). *3D Meshing* is automatically loaded with the landform files that requires it.

5.4.1. 3D Dam Surface

- **Meshing Menu, Dam:** This option (shown in Figure 26) defines a Dam landform over any given surface.

The image shows a software dialog box titled "3D Dam". It features a blue header bar with a logo on the left and window control buttons (minimize, maximize, close) on the right. The main area is light blue and contains several controls: an "Open Mesh File" button followed by a text input field; three radio buttons labeled "1 Segment", "2 Segments", and "3 Segments", with "1 Segment" selected; input fields for "Dam Crest Elevation", "Dam Crest Width", "Upstream Slope", and "Downstream Slope"; a grid of eight input fields for coordinates (X1, X2, X3, X4, Y1, Y2, Y3, Y4) with a blue "P" button to the right; a "Snap to GP" checkbox; and "Generate" and "Reset" buttons at the bottom right.

Figure 26: 3D Surfaces Menu – Dam.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the dam will be created.
- **1 Segment, 2 Segments or 3 Segments options:** These options define if the dam alignment has one, two or three segments in its crest alignment.
- **Dam Crest Elevation:** Elevation of the dam crest. This is constant for the entire Dam alignment.
- **Dam Crest Width:** Width of the dam crest. This is constant for the entire Dam alignment.
- **Upstream Slope:** Slope for the upstream faces of the dam. This must be a real value. For a slope of 1:1.75 (V:H), the value entered should correspond to the horizontal component, 1.75. For a slope of or 2:1 (V:H), the value entered must be 0.5, half of the horizontal component. The upstream direction is right hand of the Dam axis vector. If your upstream direction is left hand of the global x direction, upstream and downstream slopes must be swapped, and the (Xi, Yi) couples must correspond to the downstream side.
- **Downstream Slope:** Slope for the downstream faces of the dam. This must be a real value. For a slope of 1:1.75 (V:H), the value entered should correspond to the horizontal component, 1.75. For a slope of or 2:1 (V:H), the value entered

must be 0.5, half of the horizontal component. The downstream direction is left hand of Dam axis vector. If your downstream direction is right hand of the global x direction, upstream and downstream slopes must be swapped, and the (Xi, Yi) couples must correspond to the downstream side.

- **Xi and Yi Coordinates:** Coordinates of the upstream side of the dam crest. They must be entered such that; $X1 < X2 < X3 < X4$.
 - **Hint:** Coordinates for the Dam alignment can be selected using the mouse and the *render "F"* projection (see section 6.2) or using the paste feature (below).
 - **"P":** Will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
X1 Y1
X2 Y2
X3 Y3
X4 Y4

- **Snap to GP:** This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint:** When manual input is preferred (paste option), consider disabling this option.

- **Reset:** Deletes the current coordinates.

- **Generate:** Starts the dam surface process.

5.4.2. 3D Circular Excavation Surface

- **Meshing Menu, Circular Excavation:** This option (shown in [Figure 27](#)) defines a Circular Excavation landform over any given surface.

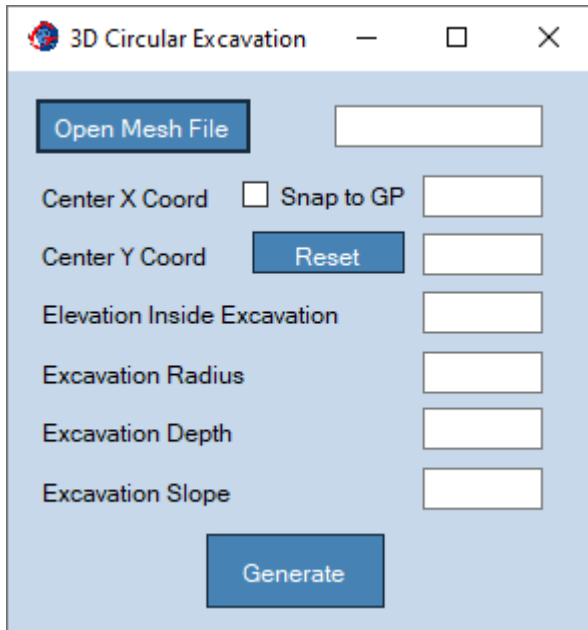


Figure 27: 3D Surfaces Menu – Circular Excavation.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the Circular Excavation will be created.
- **Center X and Y Coordinates:** These coordinates define the plan view location of the circular excavation center.
 - **Hint:** Coordinates for the circular excavation center can be selected using the mouse and the 3D render “F” projection (see section 6.2)
- **Snap to GP:** This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint:** When manual input is preferred, consider disabling this option.
- **Reset:** Deletes the current coordinates.
- **Elevation Inside Excavation:** Elevation within the area of the excavation.

- **Excavation Radius:** Radius of the excavation area.
- **Excavation Depth:** Depth of the excavation. Defines how deep the excavation is and therefore, the lateral extent of it.
- **Excavation Slope:** Slope for the excavation faces. For a slope of 1:1.75 (V:H), the value entered should correspond to the horizontal component, 1.75. For a slope of or 2:1 (V:H), the value entered must be 0.5, half of the horizontal component.
- **Generate:** Starts the circular excavation surface process.

5.4.3. 3D Vertical Fill (MSE Wall) Surface

- **Meshing Menu, Vertical Fill (MSE Wall):** This option (shown in Figure 28) defines a Vertical Fill (MSE Wall) landform over any given surface.

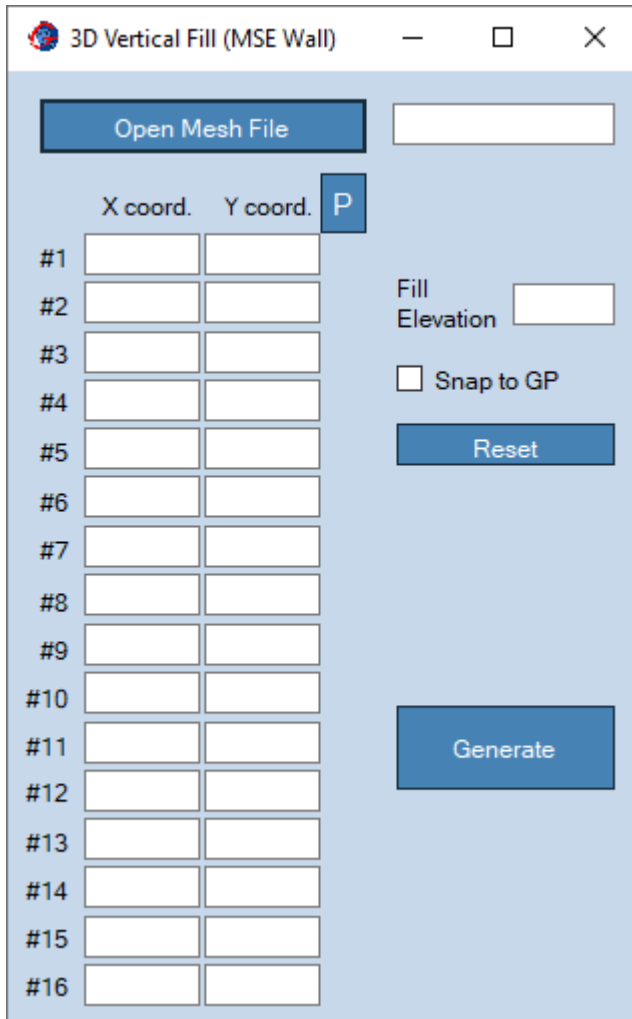


Figure 28: 3D Surfaces Menu – Vertical Fill (MSE Wall).

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the Vertical Fill will be created.
- **Xi and Yi Coordinates:** These coordinates define the plan view polygon of the Vertical Fill shape. The polygon coordinate supports up to 16 vertices and they must be defined in clockwise direction. Do not repeat the first point after the last one, there is no need to close the polygon.
 - **Hint:** Coordinates for the Vertical Fill polygon can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).

- **“P”**: will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
X1 Y1
X2 Y2
X3 Y3 ...
- **Fill Elevation**: Vertical Fill elevation at top surface.
- **Snap to GP**: This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint**: When manual input is preferred (paste option), consider disabling this option.
- **Reset**: Deletes the current coordinates.
- **Generate**: Starts the Vertical Fill surface process.

5.4.4. 3D Polygonal Fill Surface (not active with *Test License*)

- **Meshing Menu, Polygonal Fill:** This option (shown in [Figure 29](#)) defines a Polygonal Fill landform over any given surface.

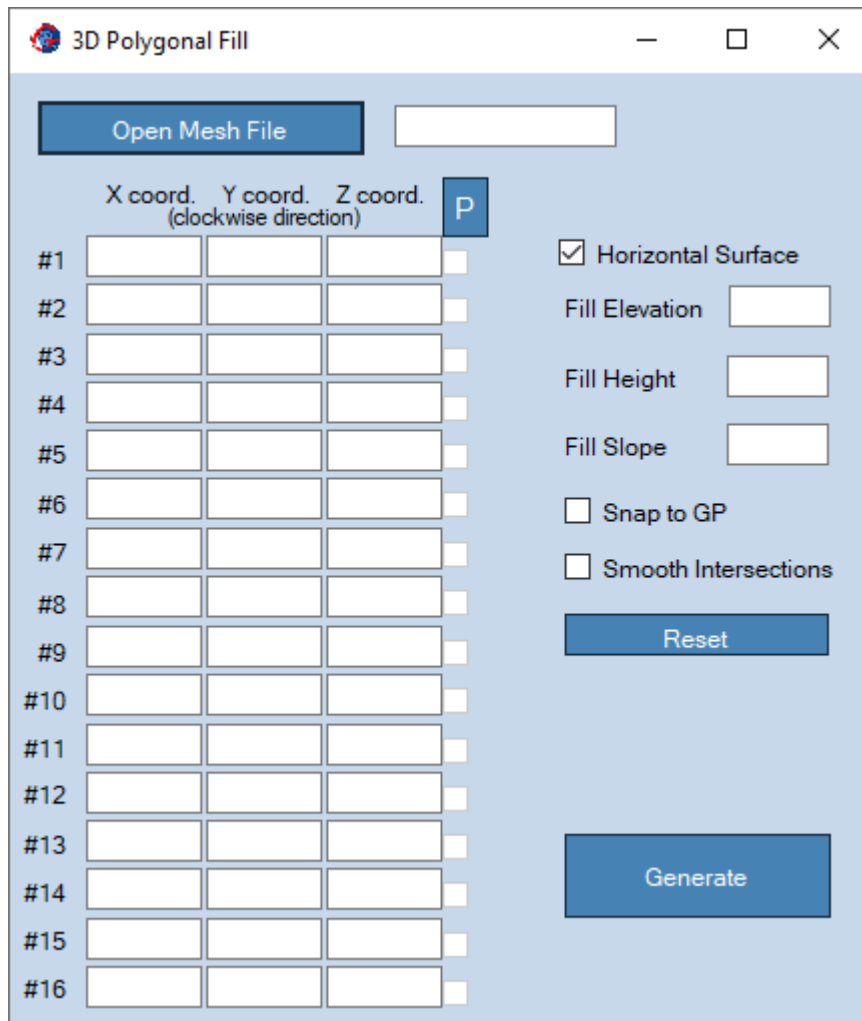


Figure 29: 3D Surfaces Menu – Polygonal Fill.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the Polygonal Fill will be created.
- **Xi, Yi and Zi Coordinates:** These coordinates define the plan view and the elevations of the main polygon. The polygon coordinates support up to 16 vertices and they must be defined in clockwise direction. Do not repeat the first point after the last one, there is no need to close the polygon.
 - **Hint:** Coordinates for the Polygonal Fill can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).

- **“P”**: will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
 - X1 Y1 Z1
 - X2 Y2 Z2
 - X3 Y3 Z3 ...
- **Horizontal surface**: When this option is selected, the polygonal fill will have a flat surface at the polygon area, with a constant elevation defined by *Fill Elevation*. Otherwise, *Geo Meshing v4.3* uses the three selected coordinates (user **MUST** select three check boxes, right next to Z coordinates) to determine an inclined plane. For this last option, Zi coordinates can be manually entered, next to each checked box.
- **Fill Elevation**: Polygonal Fill elevation at top surface.
- **Fill Height**: Polygonal Fill height from the top to the lowest point at the toe. It helps to limit the extension of the Polygonal Fill.
- **Fill Slope**: Slope for the Polygonal Fill faces. This must be a real value. For a slope of 1:1.75 (V:H), the value entered should correspond to the horizontal component, 1.75. For a slope of or 2:1 (V:H), the value entered must be 0.5, half of the horizontal component.
- **Snap to GP**: This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint**: When manual input is preferred (paste option), consider disabling this option.
- **Reset**: Deletes the current coordinates.
- **Generate**: Starts the Polygonal Fill surface process.

5.4.5. 3D Polygonal Excavation Surface (not active with *Test License*)

- **Meshing Menu, Polygonal Excavation:** This option (shown in Figure 30) defines a Polygonal Excavation landform over any given surface.

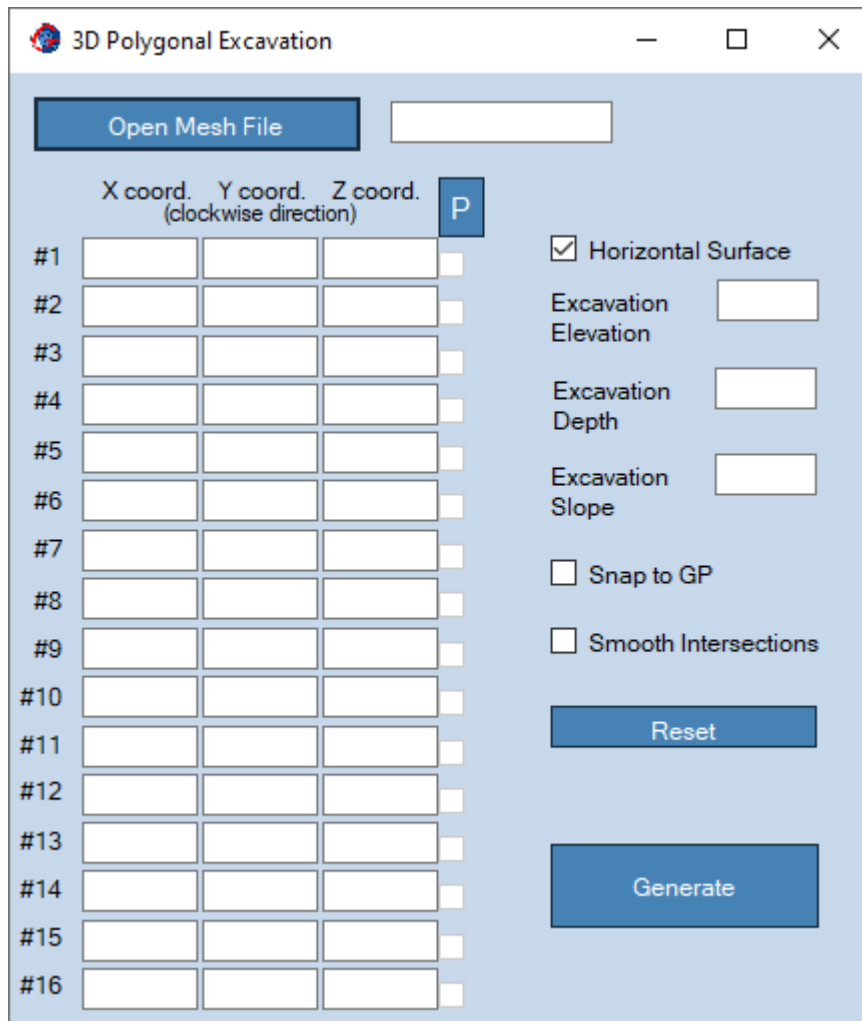


Figure 30: 3D Surfaces Menu – Polygonal Excavation.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the Polygonal Excavation will be created.
- **Xi, Yi and Zi Coordinates:** These coordinates define the plan view and the elevations of the main polygon. The polygon coordinate supports up to 16 vertices and they must be defined in clockwise direction. Do not repeat the first point after the last one, there is no need to close the polygon.
 - **Hint:** Coordinates for the Polygonal Excavation can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).

- **“P”**: it will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
 - X1 Y1 Z1
 - X2 Y2 Z2
 - X3 Y3 Z3 ...
- **Horizontal surface**: When this option is selected, the polygonal excavation will have a flat surface at the polygon area, with a constant elevation defined by *Excavation Elevation*. Otherwise, *Geo Meshing v4.3* uses the three selected coordinates (user **MUST** select three check boxes, right next to Z coordinates) to determine an inclined plane. For this last option, Zi coordinates can be manually entered, next to each checked box.
- **Excavation Elevation**: Polygonal Excavation elevation.
- **Excavation Depth**: Polygonal Excavation depth. It helps to limit the extension of the Polygonal Excavation.
- **Excavation Slope**: Slope for the Polygonal Excavation faces. This must be a real value. For a slope of 1:1.75 (V:H), the value entered should correspond to the horizontal component, 1.75. For a slope of or 2:1 (V:H), the value entered must be 0.5, half of the horizontal component.
- **Snap to GP**: This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint**: When manual input is preferred (paste option), consider disabling this option.
- **Reset**: Deletes the current coordinates.
- **Generate**: Starts the Polygonal Excavation surface process.

5.4.6. 3D Finer Mesh Surface (not active with *Test License*)

- **Meshing Menu, Finer Mesh:** This menu option (shown in [Figure 31](#)) allows for a mesh refinement with smaller elements, either over the entire model domain or over a portion of it. The new smaller elements have a quarter size of the source elements. This option does not require the execution of *3D Meshing* after generation.

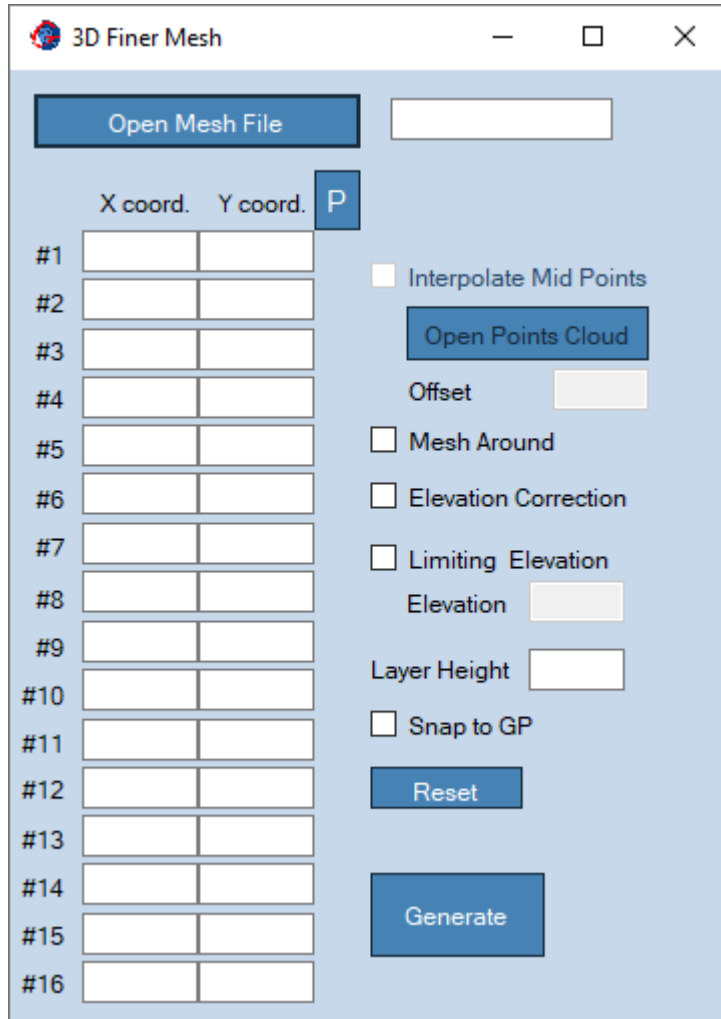


Figure 31: 3D Surfaces Menu – Finer Mesh.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the mesh refinement is applied.
- **Xi and Yi Coordinates:** These coordinates define the plan view polygon of the Finer Mesh shape. The polygon coordinates support up to 16 vertices and they must be defined in clockwise direction.

- **Hint:** Coordinates for the Finer Mesh polygon can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).
 - **“P”:** will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
 - X1 Y1
 - X2 Y2
 - X3 Y3 ...
- **Interpolate Mid Points:** This option is only activated when *Mesh Around* and *Elevation Correction* options have previously selected. Selecting this option will enable interpolation of the new generated points in the finer mesh. This option is useful when a very coarse mesh was first issued.
 - **Warning:** *Geo Meshing v4.3* do not check the correctness of the generated mesh. This option could potentially lead to distorted elements if a proper layer height is not selected. Start with the suggested value of *offset* and increase it if poor geometry is generated or decrease it if no effect is observed.
- **Mesh Around:** When this option is activated, a mesh around the finer mesh of the same size as the original is created. This option is used for minimizing the landform shape alteration.
- **Elevation Correction:** Corrects the overall elevation of the mesh. For example, if the finer mesh has a thickness of *z*, then the entire mesh model is moved downward by *z*, thus the original elevations are not affected.
- **Limiting Elevation:** Limits the extension of finer mesh procedure by a given elevation.
 - **Elevation:** Elevation value.
- **Layer Height:** Sets the Finer Mesh layer thickness.
- **Snap to GP:** This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.

- **Hint:** When manual input is preferred (paste option), consider disabling this option.
- **Reset:** Deletes the current coordinates.
- **Generate:** Starts the Finer Mesh process.

5.4.7. 3D Horizontal Finer Mesh Surface (not active with *Test License*)

- **Meshing Menu, Horizontal Finer Mesh:** This menu option (shown in [Figure 32](#)) is very similar to the previous one, but allows for horizontal orientation of the internal planes. It is useful when a portion of the mesh wants to be refined and it need to make it compatible with construction sequence. In this way, the new finer mesh can be grouped with nearby mesh and then activated them all together. The new smaller elements have a quarter size of the source elements. This option also does not require the execution of *3D Meshing* after generation.

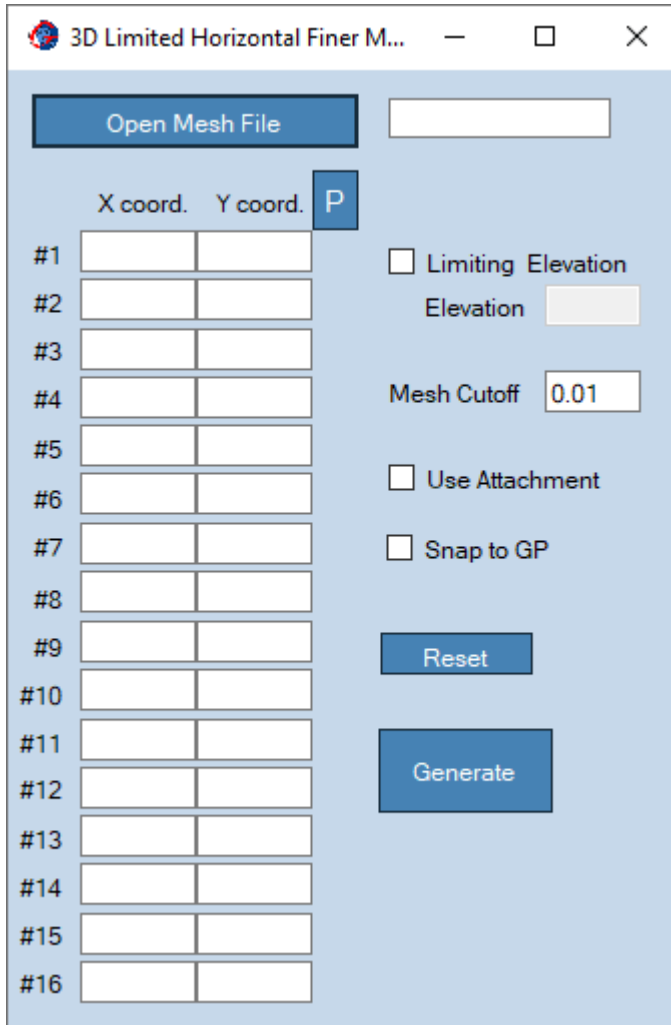


Figure 32: 3D Surfaces Menu – Horizontal Finer Mesh.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the mesh refinement is applied.

- **Xi and Yi Coordinates:** These coordinates define the plan view polygon of the Horizontal Finer Mesh shape. The polygon coordinates support up to 16 vertices and they must be defined in clockwise direction.
 - **Hint:** Coordinates for the Finer Mesh polygon can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).
 - **“P”:** will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:


```
X1 Y1
X2 Y2
X3 Y3 ...
```

- **Limiting Elevation:** Limits the extension of finer mesh procedure by a given elevation.
 - **Elevation:** Elevation value.

- **Mesh Cutoff:** Sets a maximum element height, thus small elements are not created.

- **Use Attachment:** Using this option will only produce a finer mesh (*.gmm) and it will not generate neither the *.dat file nor the *.FLAC3D file. The user must use FLAC3D© attachment command in order to make it work.

- **Snap to GP:** This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint:** When manual input is preferred (paste option), consider disabling this option.

- **Reset:** Deletes the current coordinates.

- **Generate:** Starts the Horizontal Finer Mesh process.

5.4.8. 3D Interface Surface (not active with *Test License*)

- **Meshing Menu, Interface Mesh:** This menu option (shown in Figure 33) produces a code for creating interfaces over an irregular surface created with *Geo Meshing v4.3*. This option does not require the execution of *3D Meshing* after generation.

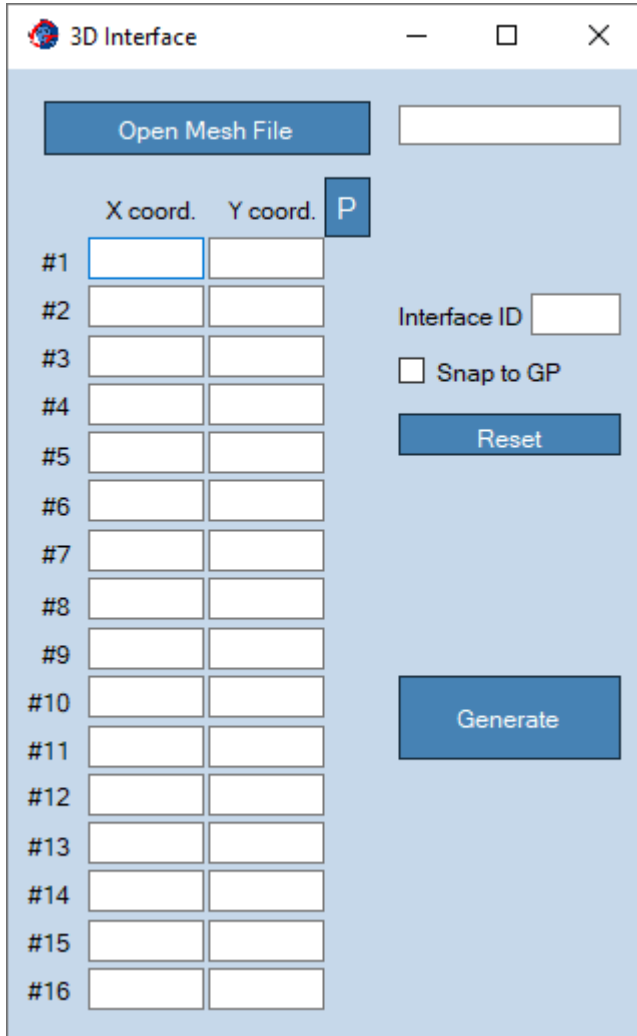


Figure 33: 3D Surfaces Menu – Interface Mesh.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the interface is applied.
- **Xi and Yi Coordinates:** These coordinates define the plan view polygon of the interface shape. The polygon coordinates support up to 16 vertices and they must be defined in clockwise direction.

- **Hint:** Coordinates for the Finer Mesh polygon can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).
- **“P”:** will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
X1 Y1
X2 Y2
X3 Y3 ...
- **Interface ID:** Defines the identification number for the interface.
- **Snap to GP:** This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint:** When manual input is preferred (paste option), consider disabling this option.
- **Reset:** Deletes the current coordinates.
- **Generate:** Starts the Interface generation process.

5.4.9. 3D Wedge Excavation Surface (not active with *Test License*)

- **Meshing Menu, Wedge Excavation Mesh:** This menu option (shown in [Figure 34](#)) defines a Wedge Excavation landform over any given surface.

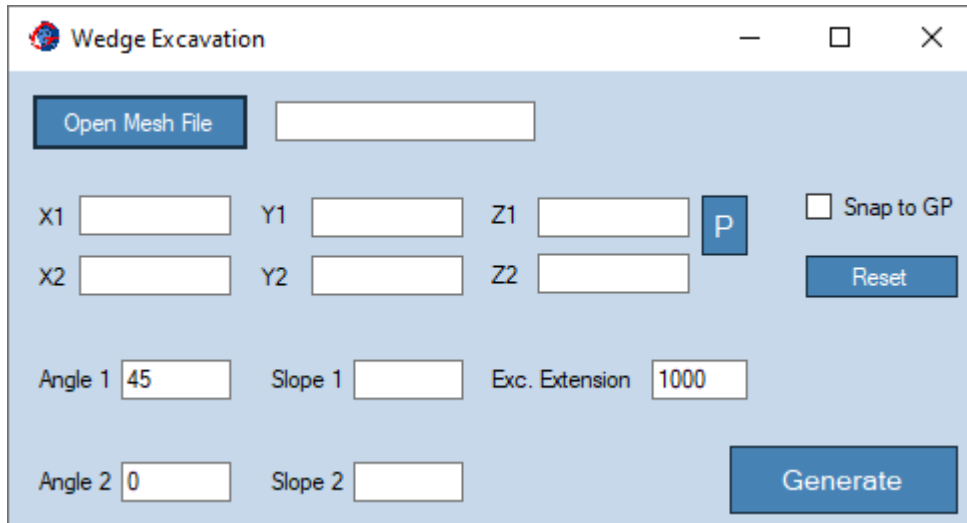


Figure 34: 3D Surfaces Menu – Wedge Excavation.

- **Open Mesh File:** Opens the *.gmm file containing the surface mesh, where the Wedge Excavation will be created.
- **Xi, Yi and Zi Coordinates:** Coordinates for defining the bottom of the Wedge Excavation.
 - **Hint:** Coordinates for the Wedge Excavation alignment can be selected using the mouse and the *render “F”* projection (see section 6.2) or using the paste feature (below).
 - **“P”:** Will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
X1 Y1 Z1
X2 Y2 Z2
- **Angle 1, Angle 2:** Intersections angles for orienting side slopes.
- **Slope 1, Slope 2:** Main slopes, perpendicular to the of Wedge Excavation bottom. This must be a real value. For a slope of 1:1.75 (V:H), the value entered should correspond to the horizontal component, 1.75. For a slope of or 2:1 (V:H), the value entered must be 0.5, half of the horizontal component.

- **Snap to GP:** This option is useful when coordinates (X,Y) are selected using the mouse. Given the unlikeliness of exactly clicking over a gridpoint (GP) coordinate, activating this option will internally correct the coordinate of the selected point to the closest GP coordinates. This option helps to create straight edges of the landform perimeter, aligned with the mesh grid.
 - **Hint:** When manual input is preferred (paste option), consider disabling this option.

- **Reset:** Deletes the current coordinates.

- **Generate:** Starts the Wedge Excavation surface process.

5.4.10. 3D Add / Subtract Surface (not active with Test License)

- **Meshing Menu, Add / Subtract Mesh:** This menu option (shown in Figure 35) operates an Addition or Subtraction between two 3D surfaces.

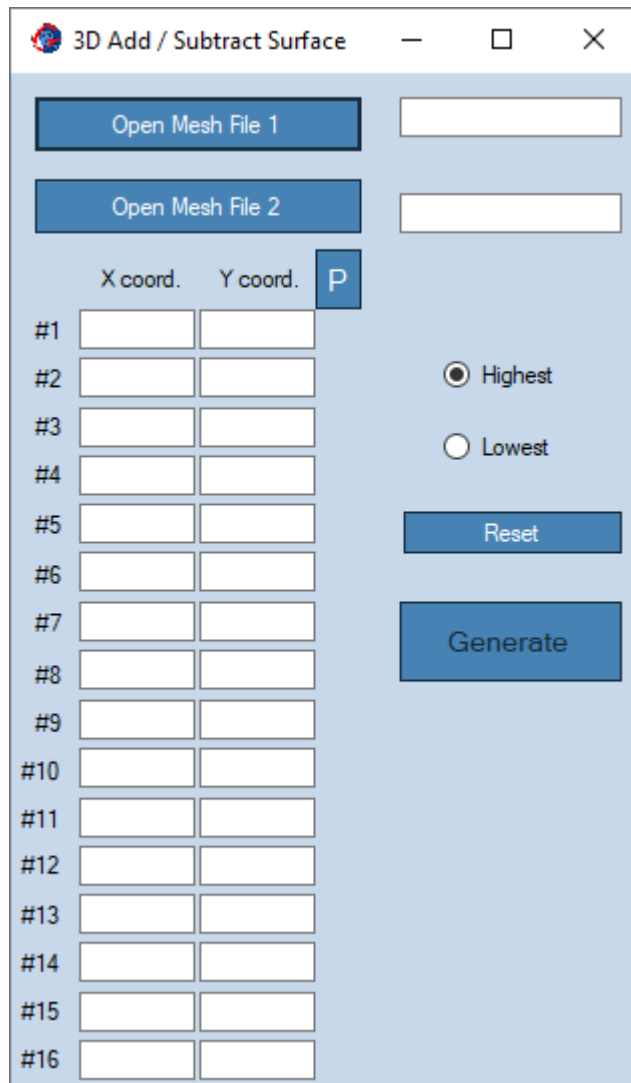


Figure 35: 3D Surfaces Menu – Add / Subtract.

- **Open Mesh File 1:** Opens the *.gmm file on which the Addition or Subtraction process is going to be performed. Additionally, this option loads the surface in the render area. *File 2* mesh can be loaded using the option *Load*, from the menu options.
- **Open Mesh File 2:** Opens the *.gmm file that it is used as reference.

- **Xi and Yi Coordinates:** These coordinates define the plan view polygon where the Addition / Subtraction applies. The polygon coordinates support up to 16 vertices and they must be defined in clockwise direction.
 - **Hint:** Coordinates for the Finer Mesh polygon can be selected using the mouse and the 3D render “F” projection (see section 6.2) or using the paste feature (below).
 - **“P”:** will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follow:
X1 Y1
X2 Y2
X3 Y3 ...

- **Highest, Lowest:** Defines if the highest or lowest elevations are kept.

- **Reset:** Deletes the current coordinates.

- **Generate:** Starts the Addition / Subtraction process.

5.5. 3D Meshing

This option executes *3D Meshing* option, as shown in [Figure 36](#).

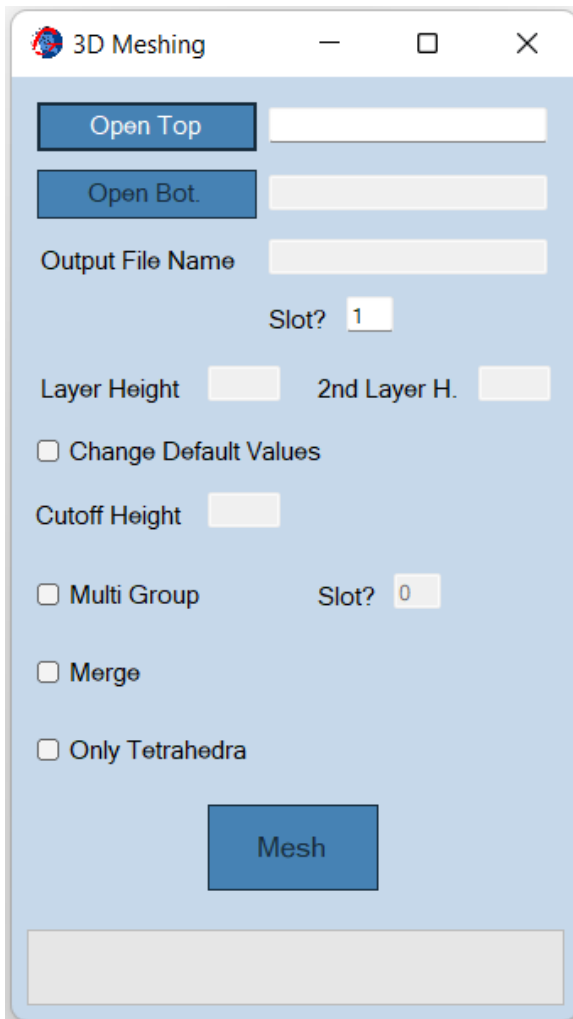


Figure 36: 3D Meshing Section.

- **Open Top Surface:** Opens the *.gmm file containing the top surface. This would normally correspond to the mesh with an overall higher elevation. After creating a landform with one of the options from section 5.4, the *Open Top* file is automatically loaded.
- **Open Bot. Surface:** Opens the *.gmm file containing the bottom surface. This would normally correspond to the mesh with an overall lower elevation. After creating a landform with one of the options from section 5.4, the *Open Bot.* file is automatically loaded.

- **Output File Name:** Sets a *name* for the mesh generated. This name is used for several outputs; such as naming the group zones, naming the *.DAT file, naming the *.flac3d files, etc. At the end of the meshing process, a file with extension “*name_mesh.gmm*” will be generated. This file can be used as a base surface mesh for further meshing processes.
- **Layer Height:** This is approximately the elements height generated at each layer during the meshing process. The number of layers in a meshing process is determined by the volume contained between the top and bottom surfaces.
- **2nd Layer H.:** The second layer height is set by default equal to *Layer Height* (above). When this value is different from *Layer Height*, a transition in height is made from *Layer Height* to *2nd Layer H*. *2nd Layer H* can be manually input smaller or larger than *Layer Height*.
- **Change Default Values:** This advanced option allows the changing of the Cutoff Height and Min. Volume values.
 - **Cutoff Height:** Sets Cutoff Height threshold, thus elements with a height less than *Cutoff Height* are not created. Change the *Cutoff Height* if elements are omitted or they are too small. By default, this value is set to 1/3 of the *Layer Height*.
- **Multi Group:** This option creates sequential groups names for each layer when meshing the domain between the top and bottom surfaces. Instead of having one *name* for all layers, as provided in *Output File Name* box, the meshing algorithm will create group names for each new layer with names *name_1*, *name_2*, *name_3*, etc. This is very useful when construction sequence is needed.
- **Only Tetrahedra:** This option generates meshes using only tetrahedrons. When this option is disabled *Geo Meshing v4.3* uses the standard FLAC3D elements, i.e. bricks, wedges, pyramids and tetrahedrons. This option is useful for using the meshes in other programs than FLAC3D.
- **Merge:** This option merges 2 opposite elements to form one element. This option considerably reduces the number of elements or zones, but it keeps the number of GP (gridpoints).
- **Mesh:** Starts the 3D Meshing process.

5.6. Plan3 Strain (not active with *Test License*)

This option creates meshes for Plane Strain analysis, either in FLAC3D or in FLAC2D. The dialog window is shown in [Figure 37](#). This option uses a DXF files for creating all layers in the Plane Strain mesh.

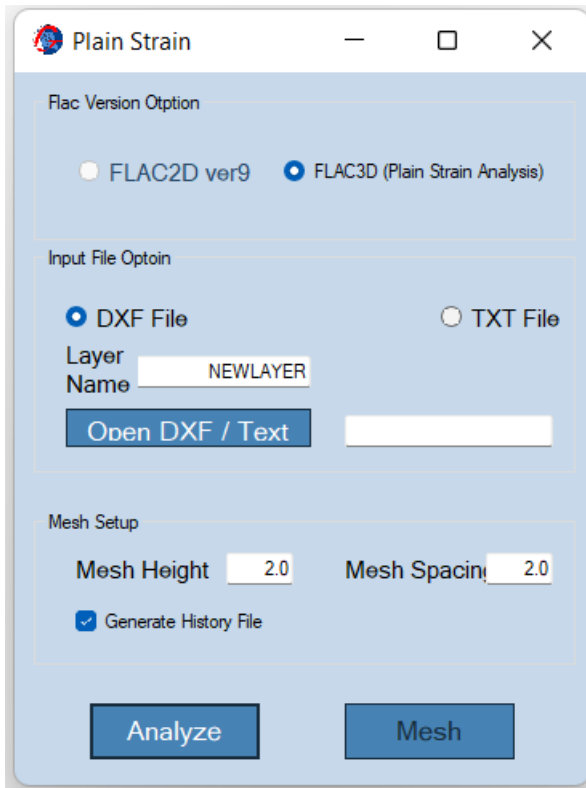


Figure 37: Plane Strain Dialog Window.

- **File Version Option:** Selects if the Plane Strain mesh will be issue for FLAC3D or for FLAC2D. In case of FLAC3D, the mesh is 1 element thick. Notice that the *FLAC2D ver9* option is currently disabled, as a stable version of FLAC2D is not available.
- **Input File Option:** Selects if the mesh information is provided by a DXF file or by a TXT file. For either case, the N layers to be generated in the mesh, must have the root name and a sequential number. For example, if the root name is “NEWLAYER”, the DXF file or TXT file should have layer definitions for NEWLAYER0, NEWLAYER1, NEWLAYER2 ... to NEWLAYERN.
 - **Layer Name:** Define the root name.
 - **Open DXF / Text:** Opens a *CAD File* or *Text File*, depending on the previous selection.
 - **Warning:** The CAD file (*.DXF) MUST be saved in R12 version, or data won't be read.

- **Mesh Setup:** Define the height and the horizontal spacing of the elements in the Plane Strain mesh.
The option *Generate History File* creates a *.gmh file that can be later edited in [Geo Meshing v4.3](#) for further editing/refining the Plane Strain mesh (see section 4.1).
- **Analyze:** Runs the analysis of either the CAD or Text file, and display the image shows in [Figure 38](#).

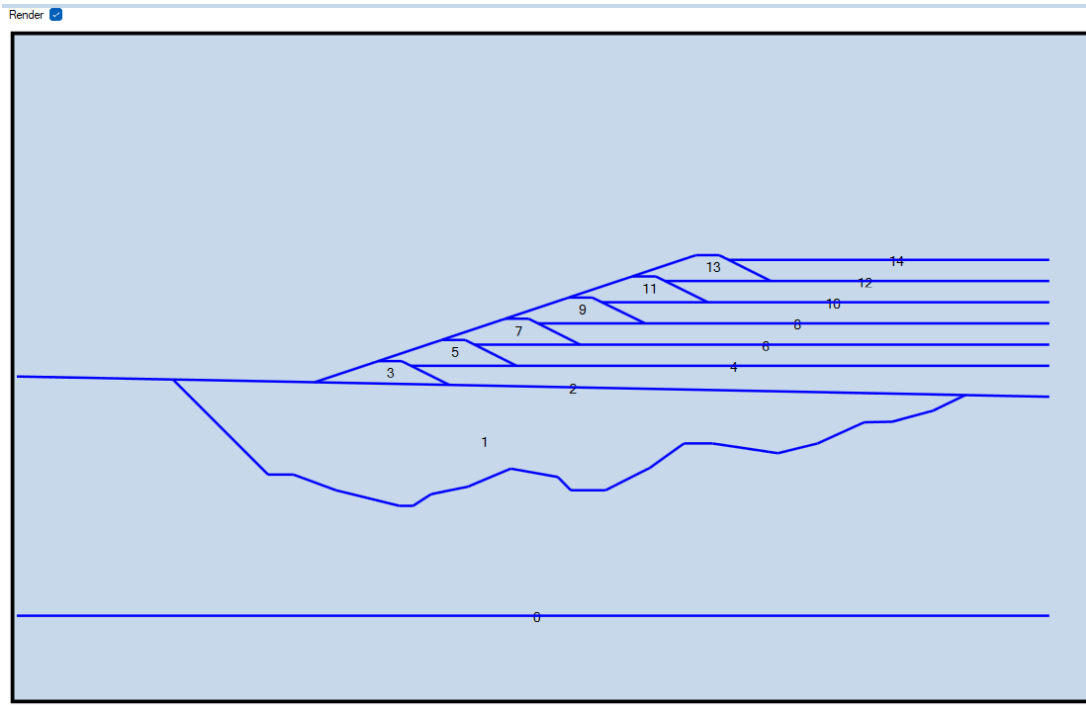


Figure 38: 2D Plain Strain Geometry in Render Area.

The render area of [Geo Meshing v4.3](#) shows the geometry along with a layering number sequence. The user must ensure that the number sequence represents a logical order of the layering, which usually follow natural geological deposition processes or construction sequence. For example, layer 3 (starter dam) cannot be defined before layer 2 (surface). Finally, notice that the lowest layer (model base) should always be numbered as 0 (zero).

- **Mesh:** Creates the Plane Strain mesh using the options defined in [Figure 37](#).

5.7. Help Menu

5.7.1. Manual Option

- **Help Menu, Manual:** This option (shown in [Figure 39](#)) opens this manual.

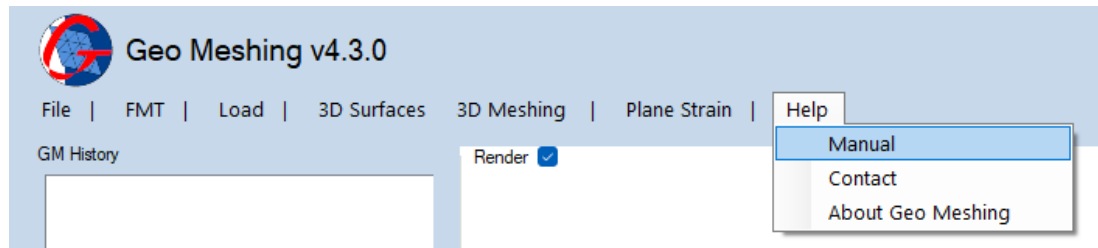


Figure 39: Help Strip Menu – Manual.

5.7.2. Contact Option

- **Help Menu, Contact:** This option (shown in [Figure 40](#)) displays support information.

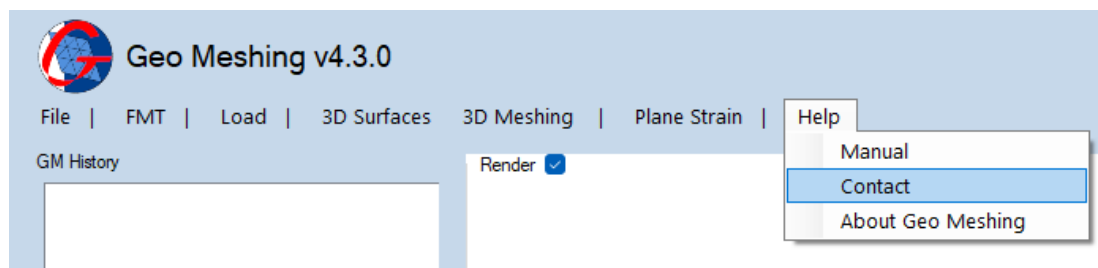


Figure 40: Help Strip Menu – Contact.

5.7.3. About Geo Meshing Option

- **Help Menu, About Geo Meshing:** This option (shown in [Figure 41](#)) displays *Geo Meshing v4.3* information along with the *Geo Meshing v4.3* code (See section 3.3 for more information).

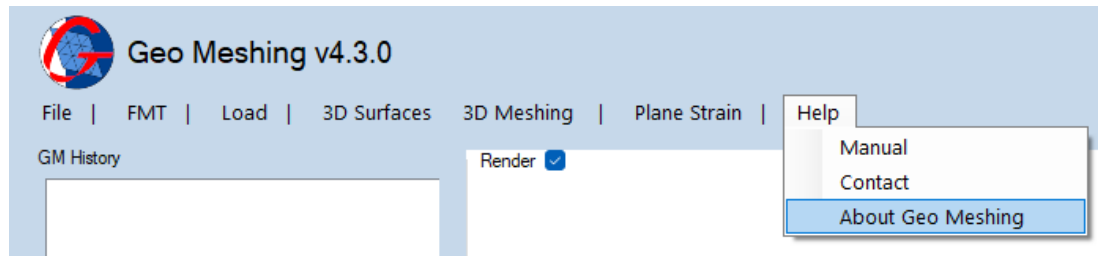


Figure 41: Help Strip Menu – About Geo Meshing.




6. Render







The *Render* section is a built-in window in *Geo Meshing v4.3*. It is a basic graphical user interface (GUI), that serves the purpose of visualizing any generated landform and 2D geometries. It also serves for GUI for input coordinates.

6.1. 3D Render Controls



Figure 42: 3D Render Section – Surface Mode.

-  Changes the model projection to top flat view.
-  Changes the model projection to front perspective view.
-  Changes the model projection to front perspective view. Rotated 90° clockwise from P1

-  Zoom in the model.
-  Zoom out the model.
-  Refreshes the model. Useful for deleting coordinate marks.
-  Alternates between activating/deactivating the mesh over the surface.
 -   Indicates whether the mesh mode is active.


6.2. Plane Strain Render Controls

There are no controls in the Plane Strain mode. If the render display is not adequate, adjust the screen size from the lower right corner.

6.3. Selecting Coordinates with Mouse

The follow procedure details how coordinates can be selected with the mouse.

- Select one of the options of section 5.4.
 - **Hint:** position the dialog window to the right of the *3D Render* area, so one can see both the mesh and dialog window.
- Use the *load* option described in section 5.3, for loading the desire *Geo Meshing v4.3* (*.gmm) mesh.

- Click  for a better model view.
- Click over the surface model.

As the user clicks on the model, a mark is displayed for reference. The first mark of the sequence is red, and the remaining marks are blue. As the marks are displayed in the *3D Render* area, the coordinates are recorded in the surface/landform dialog windows loaded in the first step.

Hint: position the dialog window to the right of the *3D Render* area, so one can see both, the mesh and dialog window.

7. FLAC3D® and FLAC® Integration

FLAC3D® and FLAC® integration is direct and effortless. When the *3D Meshing* or *Finer* options are used, a file named *name.dat* is created in the working folder. *Name* is the “Output File Name” described in section 0. When the *2D Meshing* option is used, a file named *ini_mesh.dat* is created in the working folder.

In order to import a mesh into FLAC3D® or FLAC®, select the option *call data file* from the console icon menu or just type *call name.dat* from the console command line. When generating 3D meshes, FLAC3D® will automatically and seamlessly import all grid points and zones, then it will number and group them according to the names provided in [Geo Meshing v4.3](#). In addition, the output generated by [Geo Meshing v4.3](#) is optimized for reducing the number of grid points and zones, thus Flac3D® will only merge existing grid points with imported ones when applicable. The same is true for faces on existing zones.

Again, when using 3D meshing options, and one of the two finer meshes are generated, a file called *finer_source.dat* or *hfiner_source.dat* will be created in the working folder. *Source* represent the name of the *.gmm file over where the refinement was performed. A prefix *finer_* or *hfiner_* is added to the *source* file every time a finer command is issued, thus if one performs two consecutive refinements, two meshes will be generated in the working folder; for the first refinement [Geo Meshing v4.3](#) will generate a file called *finer_source.dat* (*hfiner_source.dat*), and for the second refinement [Geo Meshing v4.3](#) will generate a file called *finer_finer_source.dat* (*hfiner_hfiner_source.dat*). Each file calls one mesh refinement.

8. Trouble Shooting

- License file is not working.
 - Get a new license file using the right *Geo Meshing v4.3* code.
 - You may need administrator privileges to run the program. If you have administrator privileges, please configure *Geo Meshing v4.3* properties to run as administrator.
 - Try changing your system date to English format (month/day/year)

- License is expired.
 - You can acquire a new license file for your *Geo Meshing v4.3* by visiting *Geo Meshing* webpage (www.geomeshing.com).

- If a DOS® window is displayed, with the message “wrong number of input parameters...”.
 - Check your folder and files hosting the project. No spaces are allowed in folder and file names. This issue has been solved in the latest *Geo Meshing* version, but still might persist depending on the folder/file name.

- Graphics errors are displayed (ghost lines).
 - The advance graphical interface might not work in all environments, try updating video drivers. In addition, ghost lines may appear if remote desktop connection is used.

- Error code c0000005.
 - Video card do not support advance graphics developed in WPF.

9. Examples

9.1. Using Geo Meshing for FLAC3D/FLAC2D – Example 1 (Plane Strain models)

This example shows how to generate a plane strain mesh of an upstream dam raise and its foundation. Figure 43 shows the sketch of the CAD drawing representing the dam and its foundation. The layers in the CAD drawing need to be defined as explained in section 5.6. Figure 44 shows the layer definition for this example.

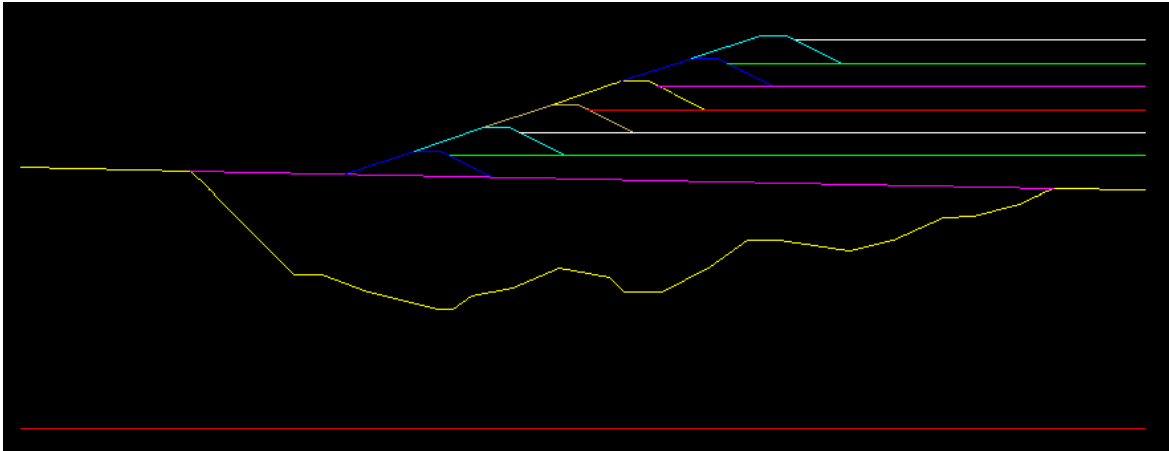


Figure 43: Dam CAD drawing.

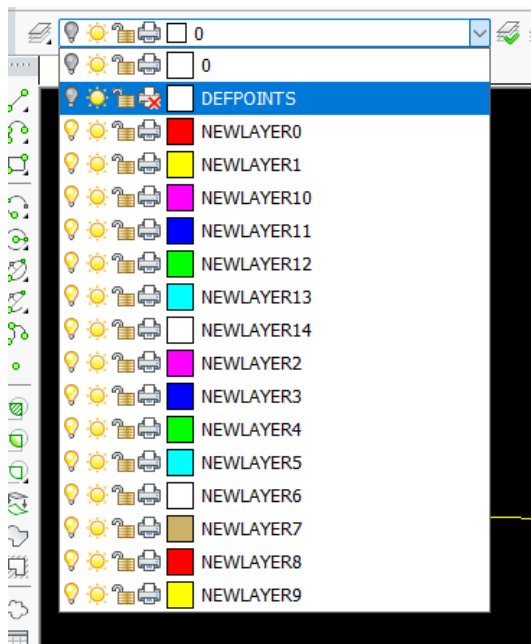


Figure 44: CAD drawing layers.

In the following steps, a red box will indicate the selected option used in *Geo Meshing v4.3* in the example.

Select *Plane Strain* as shown in [Figure 45](#).

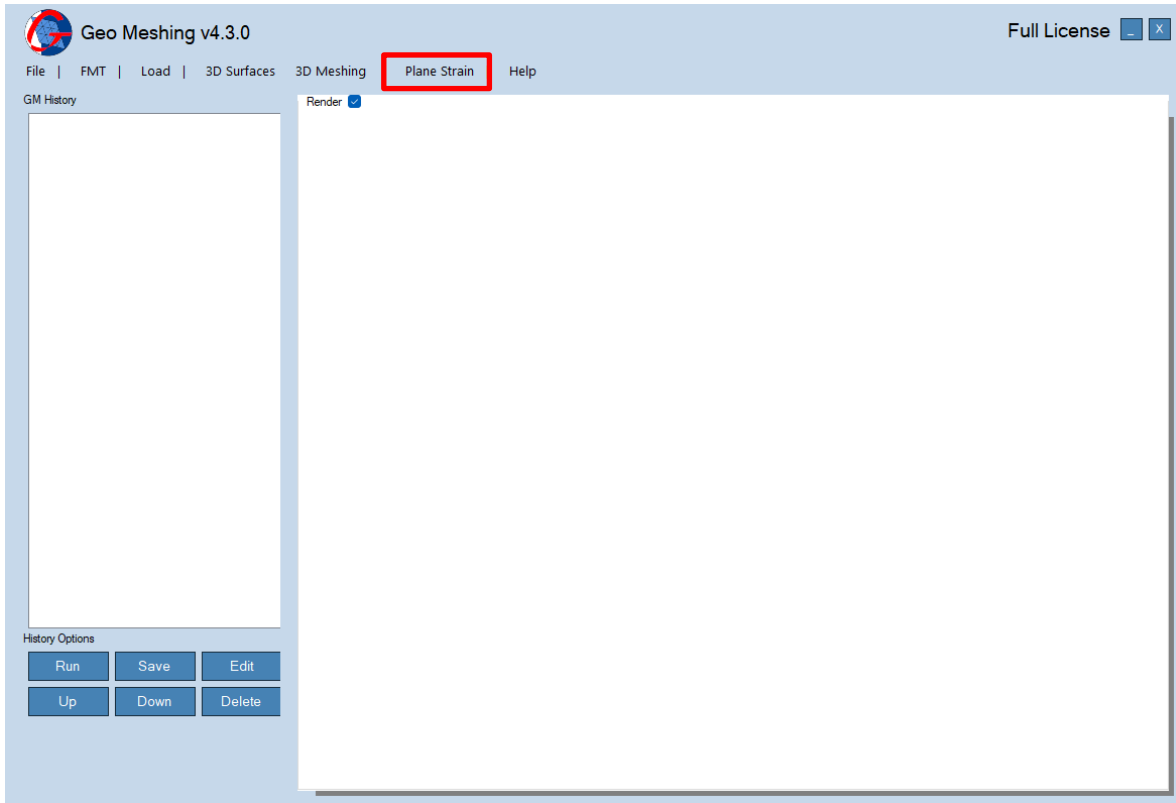


Figure 45: Geo Meshing – Plane Strain.

if the history file has not been defined, the history file dialog window will appear as shown in [Figure 46](#). Select cancel as shown in this figure. Notice: the events of Plane Strain meshing are not recorded in the history file.

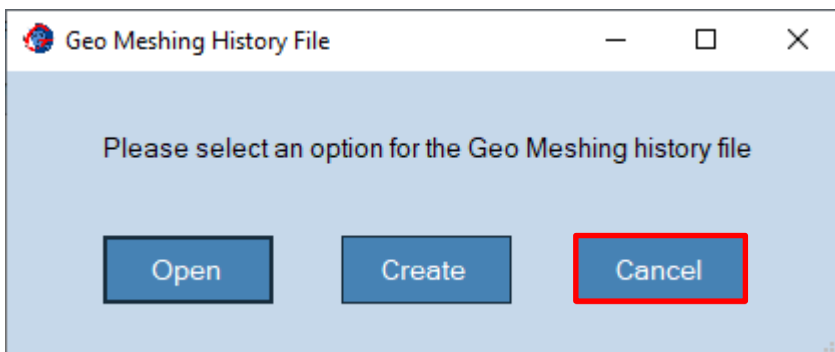


Figure 46: History file dialog window.

After selecting Cancel, the dialog window shown in [Figure 47](#) will appear. Notice that by default *FLAC3D* option is selected and *FLAC2D ver9* option cannot be selected. This last is due to *FLAC2D* version is still under development and a final nomenclature has not been decided.

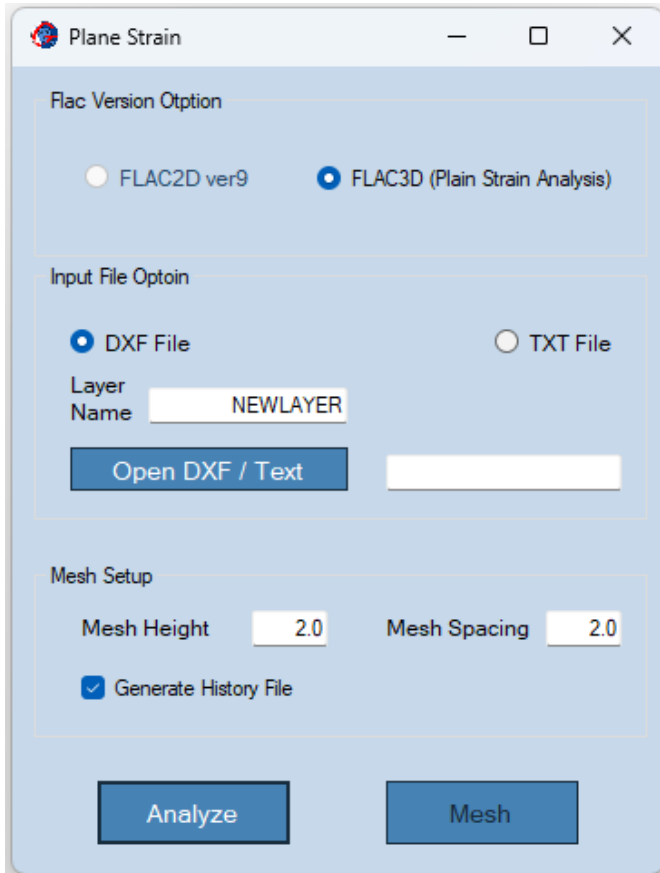


Figure 47: Plane Strain option – Open.

Select DXF file (default option), keep the layer name and click *Open DXF / Text*. The dialog window of [Figure 48](#) will pop up. Navigate to the folder where Example1.dxf file is located (C:\Program Files (x86)\Geo Meshing\Examples\Example 1) and select the file. Click open as it is shown in [Figure 48](#).

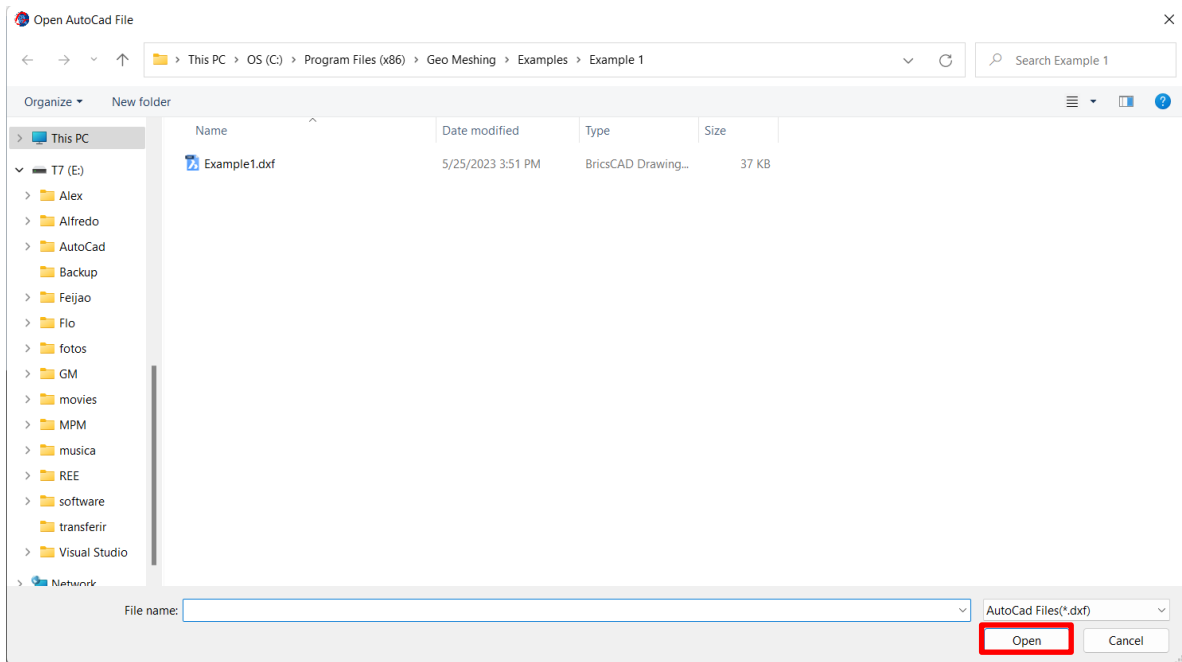


Figure 48: Open CAD file dialog window.

As shown in [Figure 49](#), the name of the DXF file will be displayed next to the *Open DXF / Text* button. Enter the mesh height and spacing as shown in [Figure 49](#). Make sure *Generate History File* is selected and click *Analyze*. The DXF is loaded into [Geo Meshing v4.3](#) as shown in [Figure 50](#). If needed, adjust the window size by dragging the windows' lower right corner. Notice that [Figure 50](#) shows the DXF geometry along with a layering number sequence. The user must ensure that the number sequence represents a logical order of the layering, which usually follows natural geological deposition processes or construction sequence. For example, layer 3 (starter dam) cannot be defined before layer 2 (surface). Finally, notice that the lowest layer (model base) should always be numbered as 0 (zero).

Finally click Mesh, this will start the meshing process.

For this example, the default values of the mesh height and spacing were modified to achieve a specific mesh density. Experimentation with different values in those boxes is encouraged. By trying different values, it will show the power and flexibility of [Geo Meshing v4.3](#).

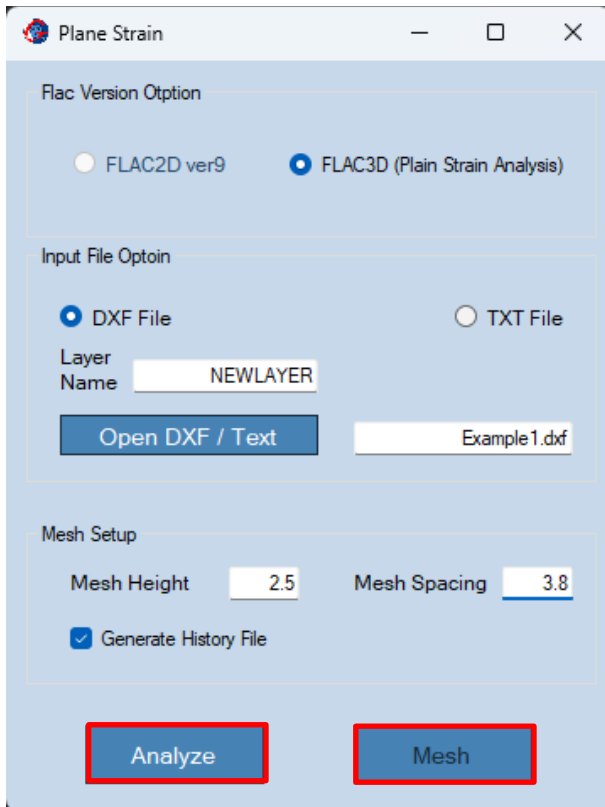


Figure 49: Plane Strain option – Analyze.

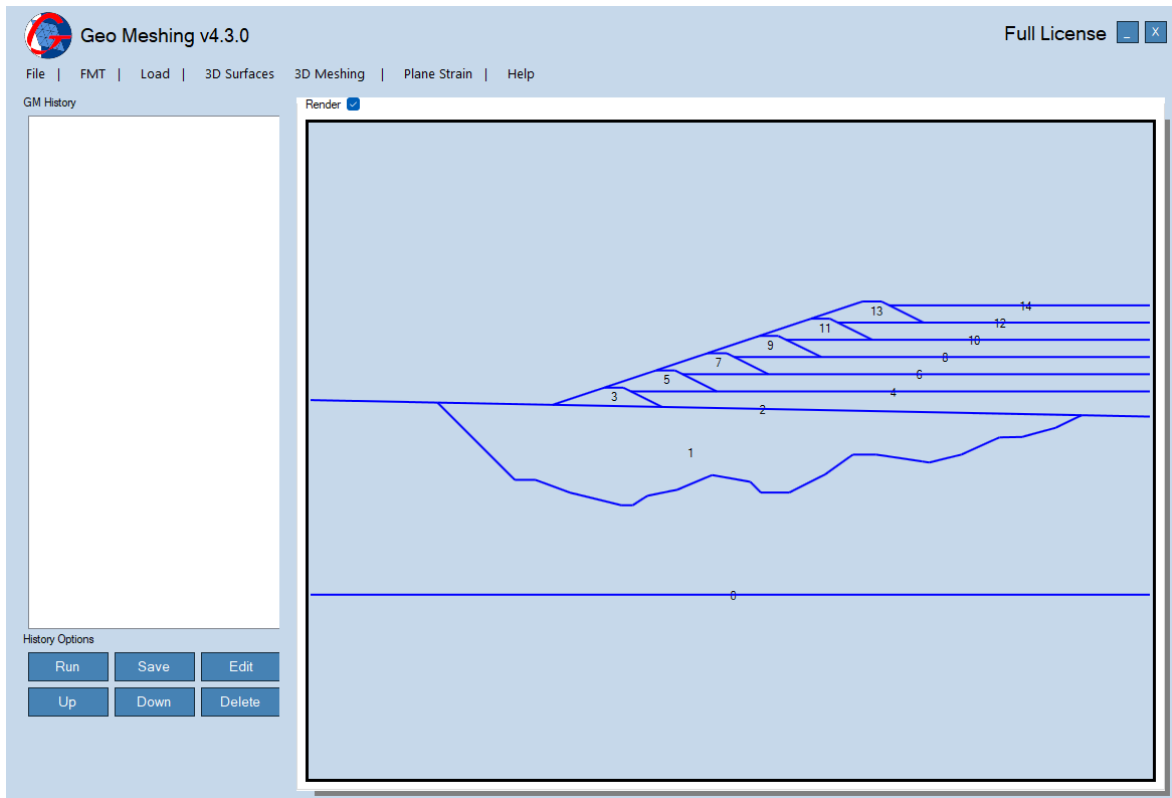


Figure 50: Geo Meshing – 2D wireframe render.

A file called *call_all.dat* will be created In the *Example 1* folder. This file can be loaded in FLAC3D using the option *call* (📄) under FLAC3D main menu. The Plane Strain model is shown in [Figure 51](#).

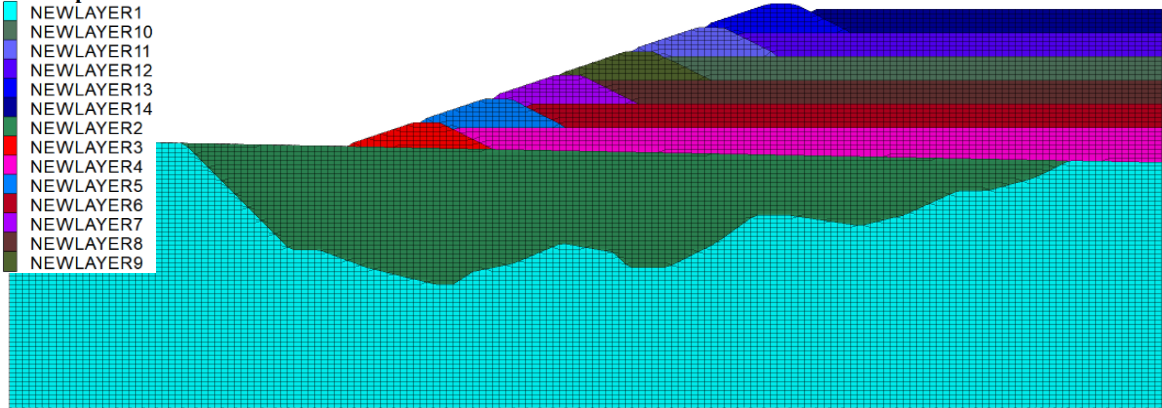


Figure 51: FLAC3D Plane Strain model.

9.2. Edit History – Example 1 continuation (Plane Strain Model)

Most of the events generated in *Geo Meshing v4.3* are recorded in the history file, when the file has been properly setup (section 4.1). This file can be edited in *Geo Meshing v4.3*, for further tailoring /editing meshes. In the following example, the previously generated Plane Strain mesh (section 9.1) is going to be edited. This same procedure can be applied to full 3D meshes.

Open *Geo Meshing v4.3* and click the *Edit* button on the lower left area of the main screen (Figure 52). The *Geo Meshing History File* windows will pop up and select open, as shown in Figure 52. Navigate to where Example1.dxf file is located (C:\Program Files (x86)\Geo Meshing\Examples\Example 1), select the file and click open as shown in Figure 53.

Figure 54 shows the history file loaded in *Geo Meshing v4.3*.

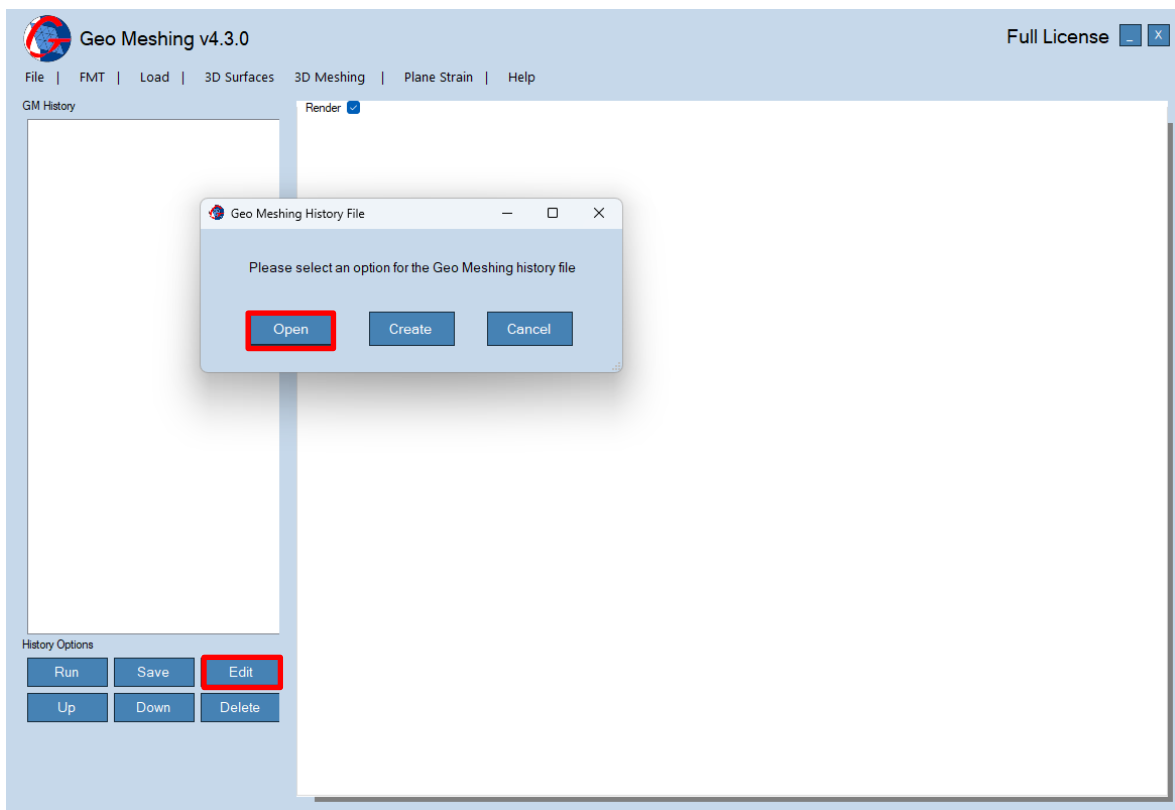


Figure 52: History Edit.

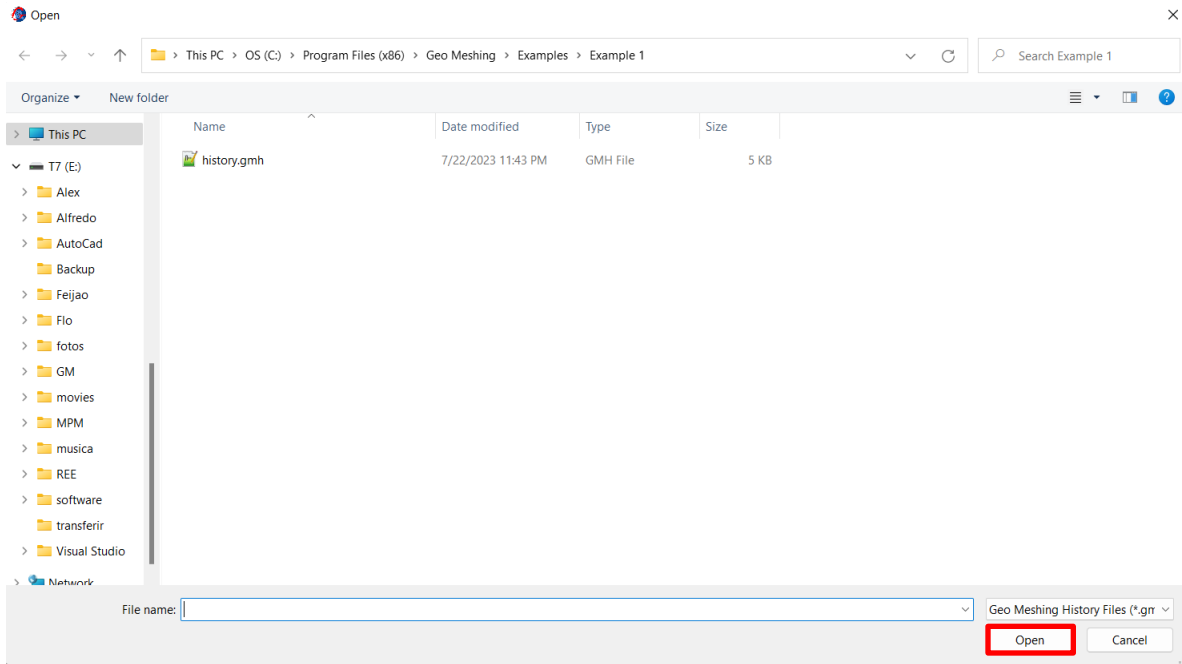


Figure 53: Open Geo Meshing History File.

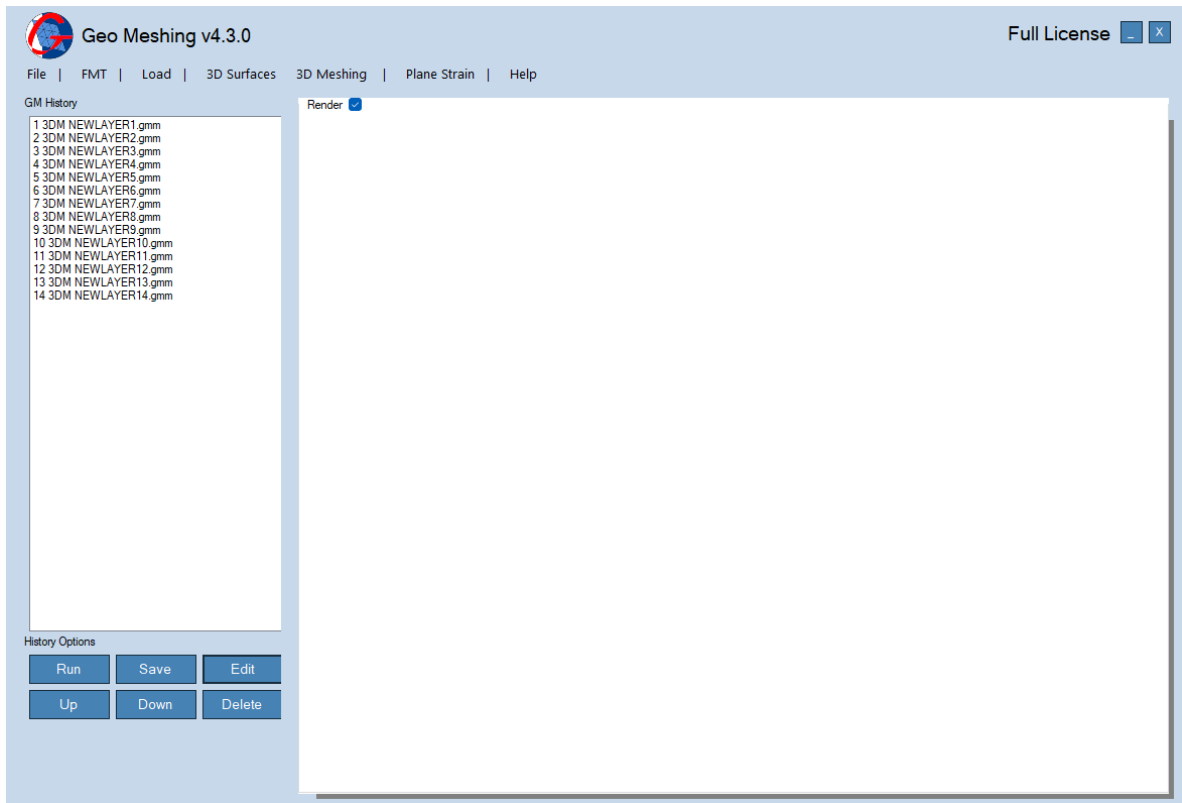


Figure 54: Geo Meshing History File Loaded.

Click again the *Edit* button and the windows shown in [Figure 55](#) will appear. Notice that there are 14 events recorded in the history, corresponding to the previously 14 layers created in the Plane Strain mesh in section 9.1.

The objective in this example is making the foundation layers coarser at the bottom of the model, and finer as the meshing process approaches to the surface. To do that, the layer heights for event 1 and event 2 will be changed to 8 and 4 units respectively ([Figure 56](#)). Notice that the 2nd Layer Height won't be changed, otherwise the element at the foundation surface won't be the same size as the rest of the upper model.

Event	Layer Height	Cutoff Height	Multi Group	Merge	2nd Layer H.	Change Cut off	Only Tetrahedra
1 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
2 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
3 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
4 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
5 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
6 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
7 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
8 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
9 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
10 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
11 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
12 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
13 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	
14 3DM, 3D Meshing	2.5	1.6666	true	false	2.5	false	

Figure 55: Geo Meshing Edit Windows.

1 3DM, 3D Meshing	Layer Height	8	Cutoff Height	1.6666
2 3DM, 3D Meshing	Layer Height	4	Cutoff Height	1.6666
3 3DM, 3D Meshing	Layer Height	2.5	Cutoff Height	1.6666

Figure 56: Layer Height Edited.

Click *Save*, the *Edit History* window will close. Now select the first event in *GM History* box and click *Run*. Edit the second box in the run windows that just popped up. Replace the value for 14, as shown in [Figure 57](#). This process will re-run the entire meshing algorithm with the changes just made. Click *Run*.

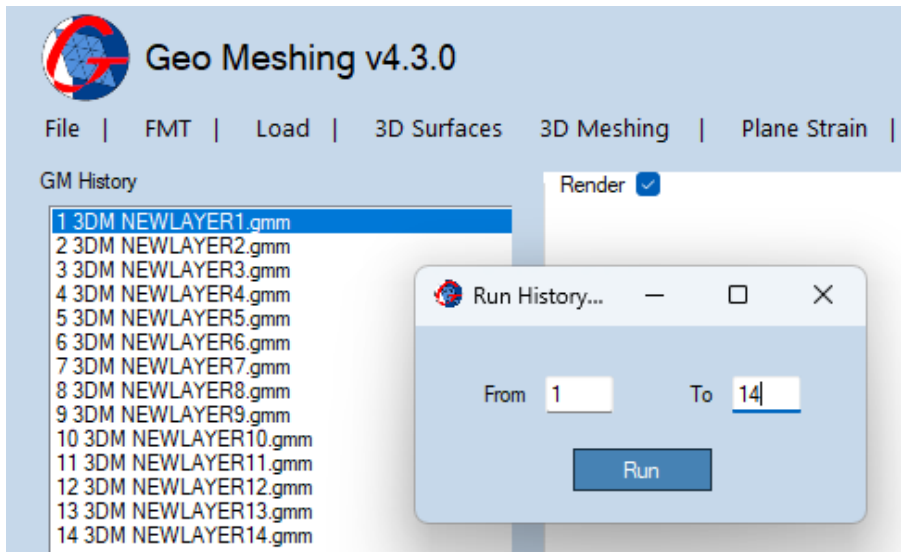


Figure 57: Run History.

Once the process has been completed, *Geo Meshing v4.3 GM History* box will display another 14 events for a total of 28 events. Do not save the history file as it will save all 28 events (Figure 58). You need to either erase the first 14 events or the last 14 events before saving the history file. To do so, just click and select the first event and then click *Delete* as many times as need to delete the first 14 events. Now you can click *Save* to save the *Geo Meshing v4.3* history file.

Figure 59 shows the full FLAC3D Plane Strain model. You can load this model following the instruction described in the previous section. Notice that the mesh is much coarser at the base, but still is of a smaller size as the foundation is approaching to the surface. The different between the edited model and the original can be observed by comparing Figure 59 with Figure 51.

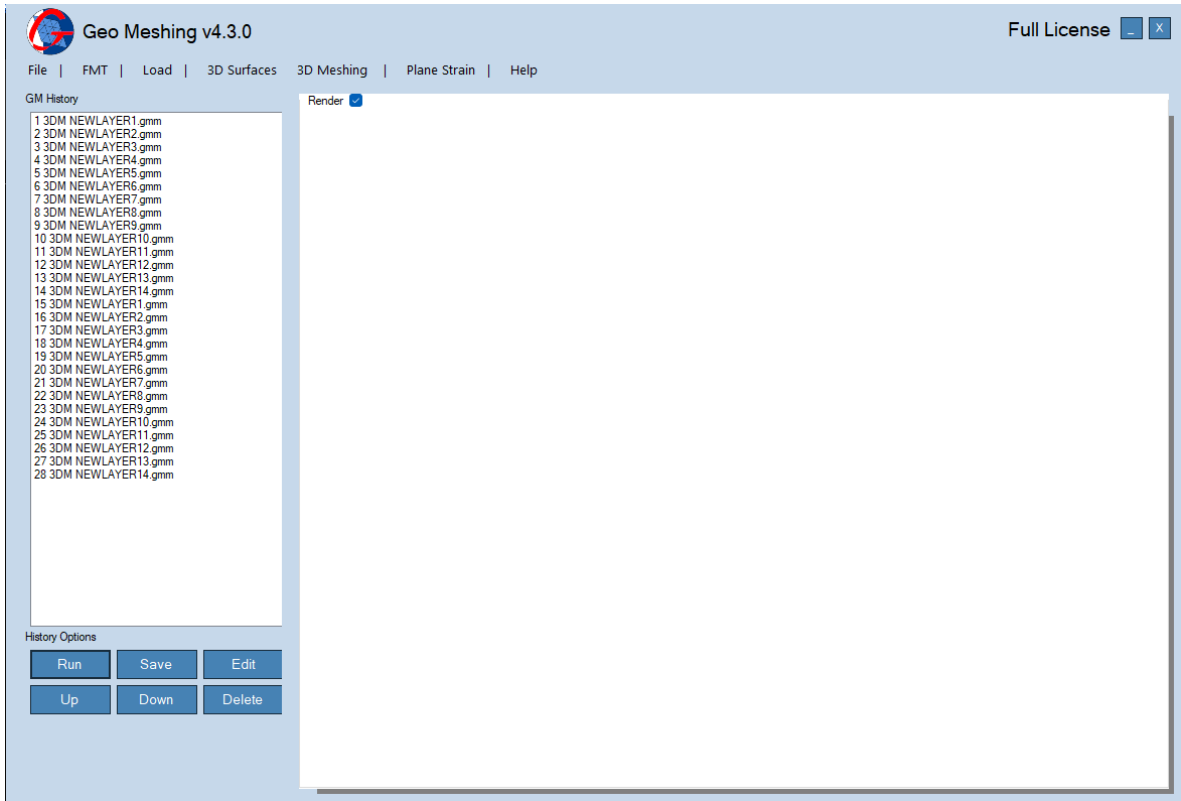


Figure 58: Geo Meshing Extended History File.

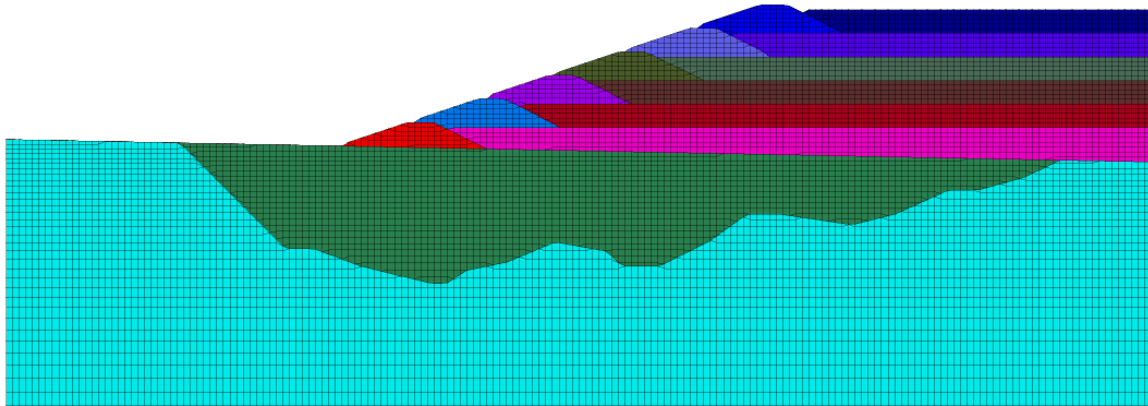


Figure 59: FLAC3D Plane Strain model. Coarser Base.

9.3. Using Geo Meshing for FLAC3D – Example 2 (3D models)

This example shows the classic *bathtub* (BT) 3D meshing model. In contrast with other meshing software, the example developed in *Geo Meshing v4.3* starts from scratch instead of an advanced 3D CAD model. Other software needs extensive pre-processing steps, where the 3D volume needs to be ready before importing into such meshing software. *Geo Meshing v4.3* is a self-contained software that can do everything in one single package.

Figure 60 shows the BT sketch. Notice the model is showing a quarter of the full BT. The model is relatively simple, but includes complex geometrical components, such as the curved plane surrounding the bottom of the BT.

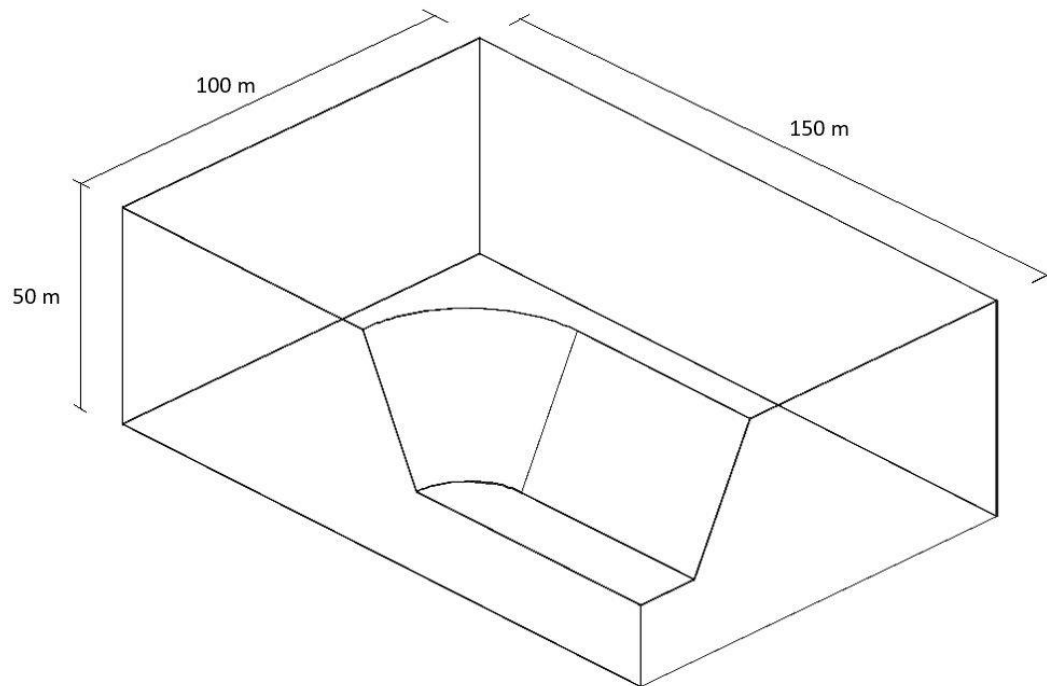


Figure 60: *Bathtub* sketch.

The essential files for creating this 3D models are located in (C:\Program Files (x86)\Geo Meshing\Examples\Example 2), but it is encouraged to try this from scratch in a different folder.

Figure 61 shows the first step, create a txt file using (for example) Notepad and input the coordinates as shown. Save the file.

This file has 3 coordinates per line. Each number represent X, Y and Z coordinates, respectively. Notice that the selection of X,Y coordinates matches the extension of the problem shown in Figure 60, assuming an origin at 0,0,0. Generally, the third coordinate is not relevant, because it is common that elevations in the base mesh are interpolated.

Although for this example, Z coordinate will be equal to the upper elevation in the sketch of [Figure 60](#).

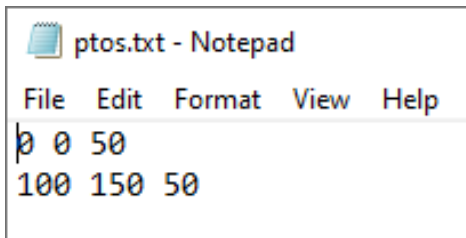



Figure 61: Notepad initial steps.

Open [Geo Meshing v4.3](#) and select *FMT*. When prompted by [Geo Meshing v4.3](#) create a history file using the suggested default name and click *Save*.

In the *FMT* window, select *Text File* option and click *Open Points Cloud*. Navigate to the folder where you saved the file of [Figure 61](#) and open it. Set the options as in [Figure 62](#) and Click *Mesh / Interpolate* button. Two files are created in the working folder, one with extension *_top.gmm* and another one with extension *_bot.gmm*.

Come back to [Geo Meshing v4.3](#) and click the option *Load* and select the file with extension *_top.gmm*. Once loaded click the  button in the render area.

Again, in [Geo Meshing v4.3](#) and under *3D surfaces* menu, select *3D polygonal excavation* surface. Arrange the windows as shown in [Figure 63](#).

Locate the mouse near $(x,y) = (0,70)$ and click. You will see a red mark in the render area and the first coordinate appears in the polygonal excavation window. Keep clicking clockwise to approximately form a quarter of a circle. You will see a series of blue marks in the render area and seven coordinates are displayed in the polygonal excavation window, as shown in [Figure 63](#).

If coordinates need to be input again, press  in the render area and the *Reset* button in the *3D polygonal excavation* surface.

Once the coordinates are ready, manually input coordinates #8 and #9 with $(26.95,0,50)$ and $(0,0,50)$, respectively. Notice that the X component of coordinate #8 must be the same value as the X component of coordinate #7. In addition, change X component of coordinate #1 to 0.0. Click *Open Mesh File* and select the file *_top.gmm*. Finally, complete the options of the *3D polygonal excavation* dialog window as shown in [Figure 64](#) and click *Generate*.

After the process is completed, use *Load* option and load the file with extension "..._top_exc.gmm". [Geo Meshing v4.3](#) will display the new surface as shown in [Figure 65](#).

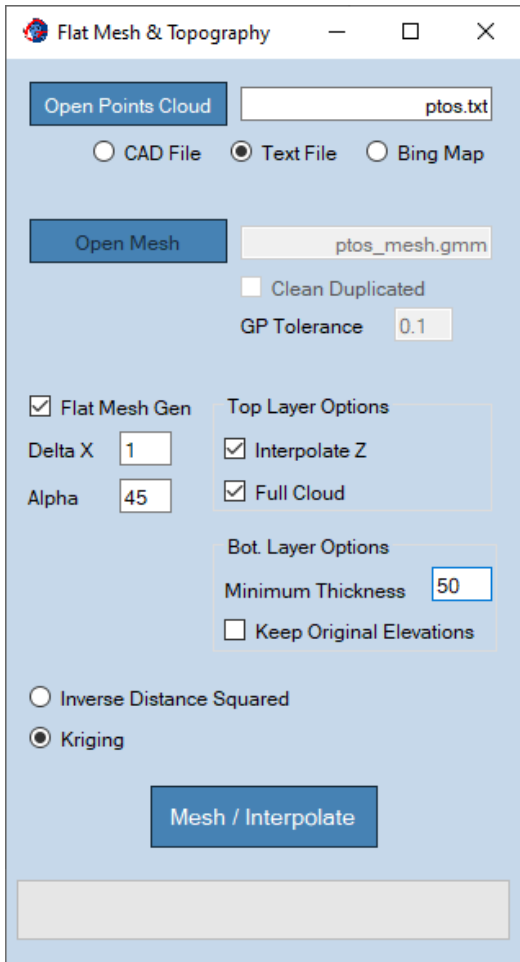


Figure 62: FMT option – Geo Meshing v4.3.

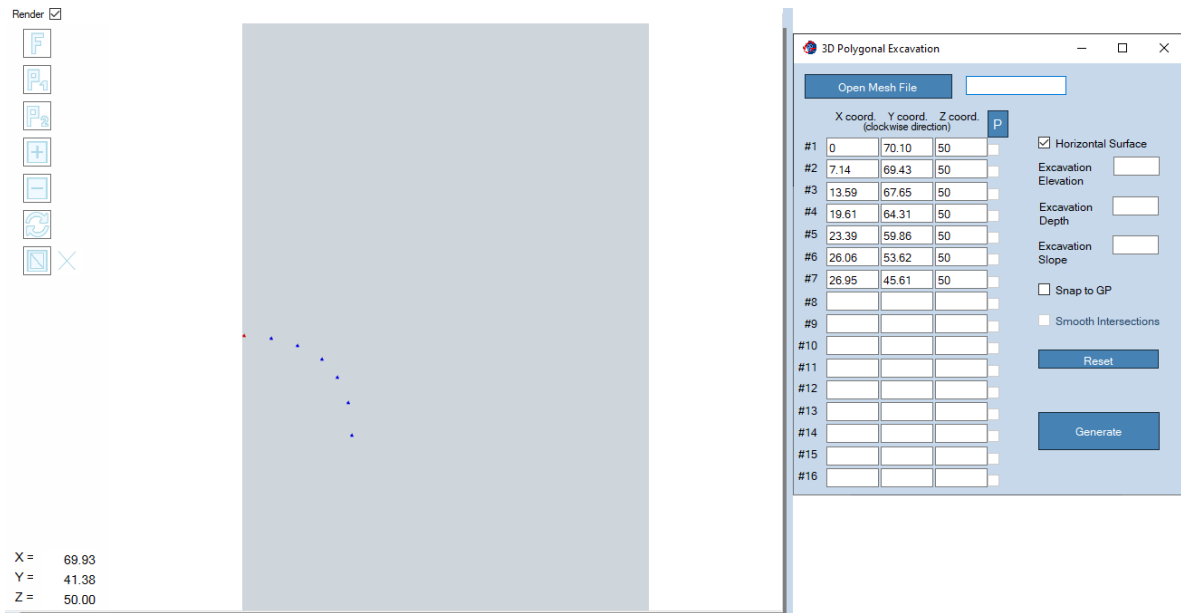


Figure 63: “F” view in render area – Geo Meshing v4.3.

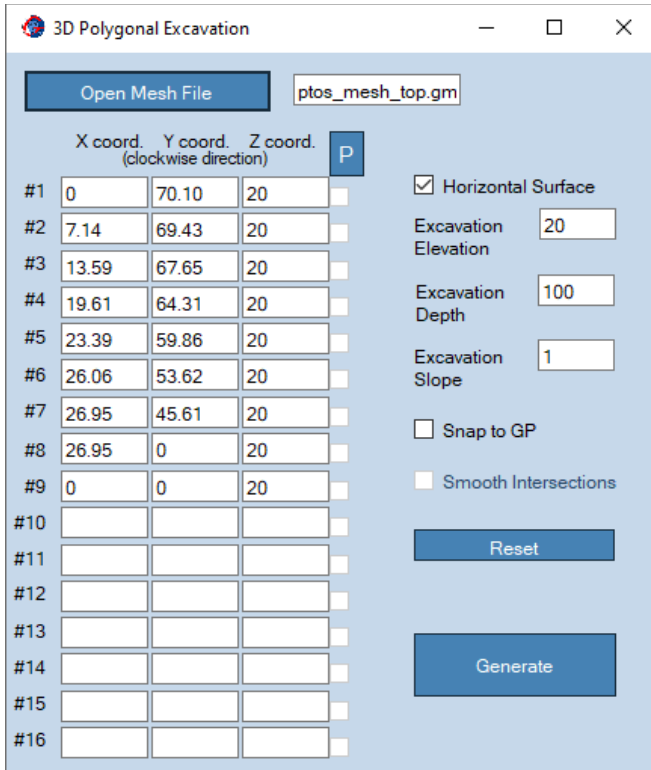


Figure 64: Polygonal excavation – Geo Meshing v4.3.

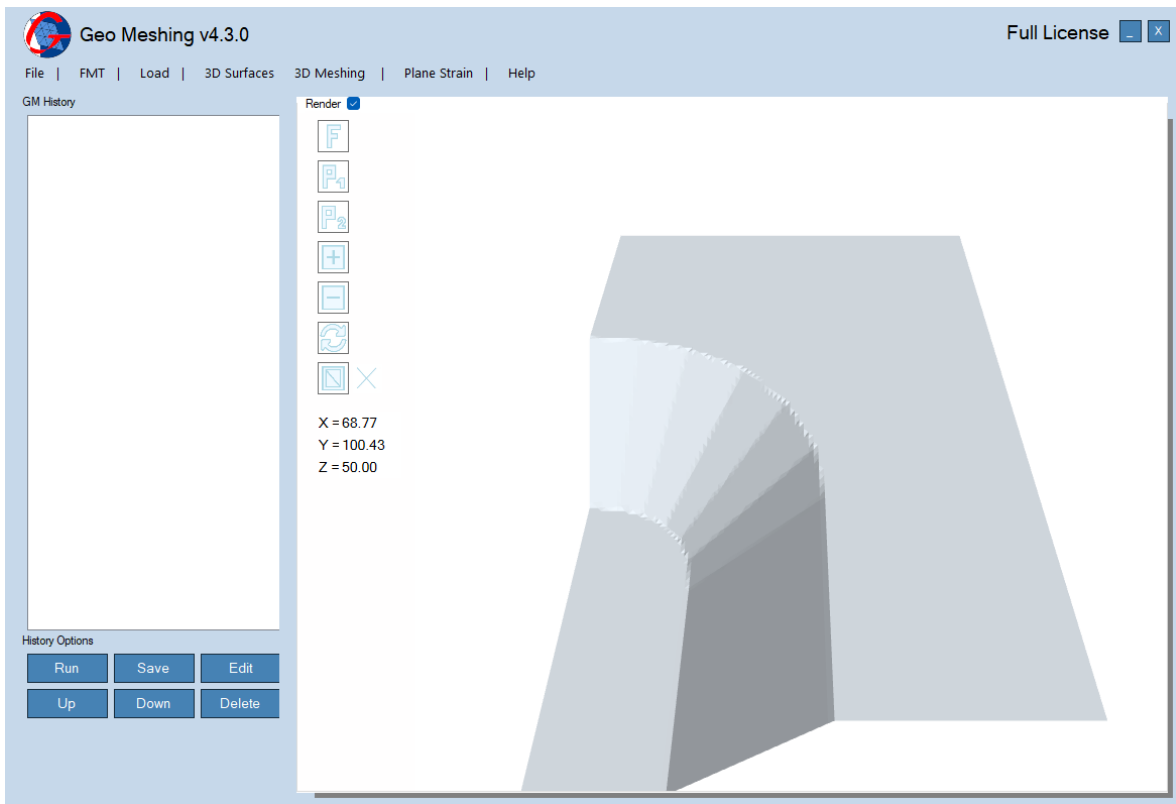


Figure 65: Geo Meshing – bathtub surface.

Congratulations, you just created in simple steps a mesh *bathtub* surface!

Now it is time to mesh it and import it into FLAC3D®. Use the *3D meshing* window (it should be open and it should look like the left image of [Figure 66](#)). By default *Geo Meshing v4.3* will load into the *3D meshing* window the surface with the higher elevation as the top surface, and the surface with the lower elevation as the bottom. Load and change the files, and complete the remaining information so the *3D meshing* window looks like the right image on of [Figure 66](#). Finally, click *Mesh*. In the working folder you will see that a *base.dat* file was created.

From FLAC3D®, call the file *base.dat* and plot the zones. The mesh should look like the one in [Figure 67](#). Notice that the height of the elements goes from 3 units at the bottom to 1 unit at the top, as specified in the *3D meshing* window.

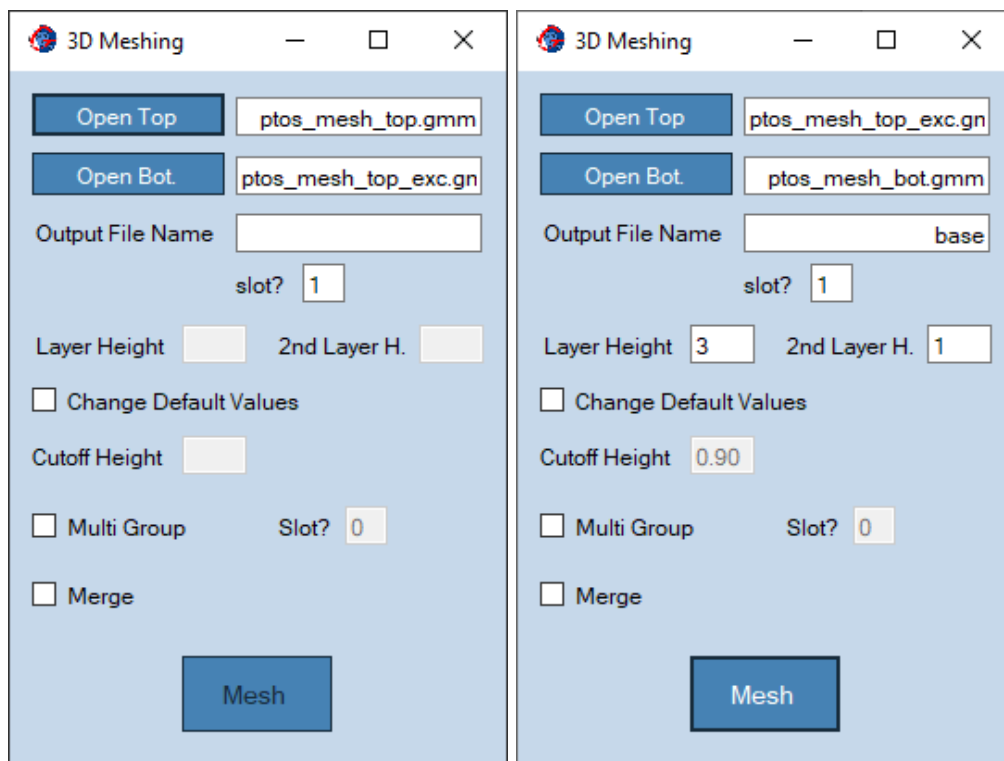


Figure 66: 3D Meshing Window – base meshing.

Usually, this example presented with other meshing tools finish here, but given the flexibility of *Geo Meshing v4.3*, further meshing can be done using a simple extra step. Go back to the *3D meshing* dialog window and change the files, complete the information as shown in [Figure 68](#) and click *Mesh*. Again, in the working folder you will see that a *bathtub.dat* file was created. Call this file from FLAC3D®. The mesh should look like the one in [Figure 69](#).

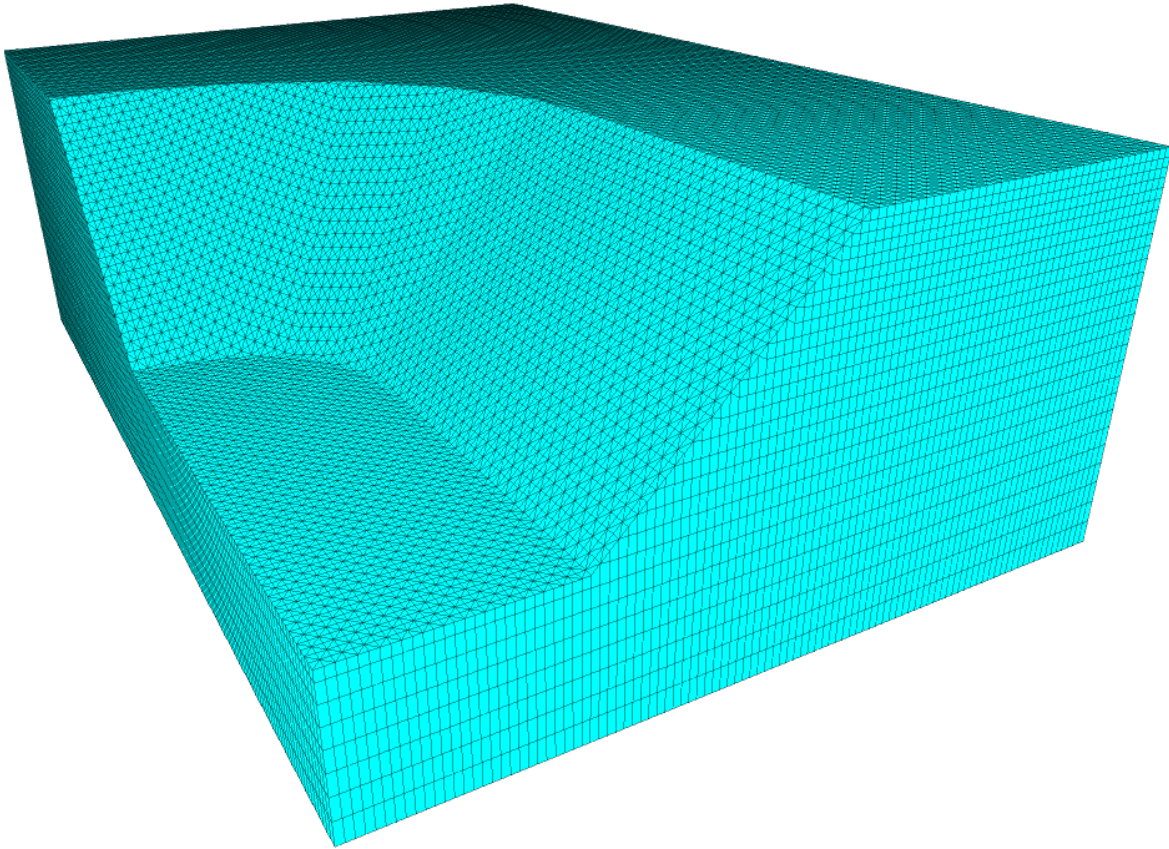


Figure 67: *Bathtub* base mesh – image from FLAC3D©.

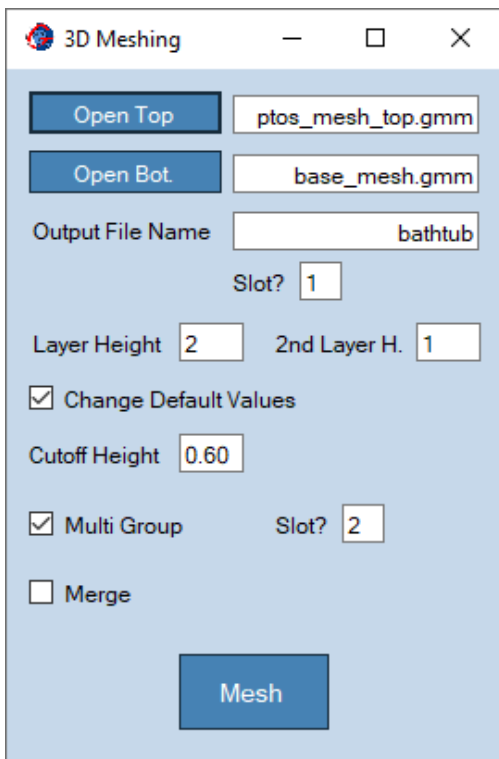


Figure 68: 3D Meshing Window – fill meshing.

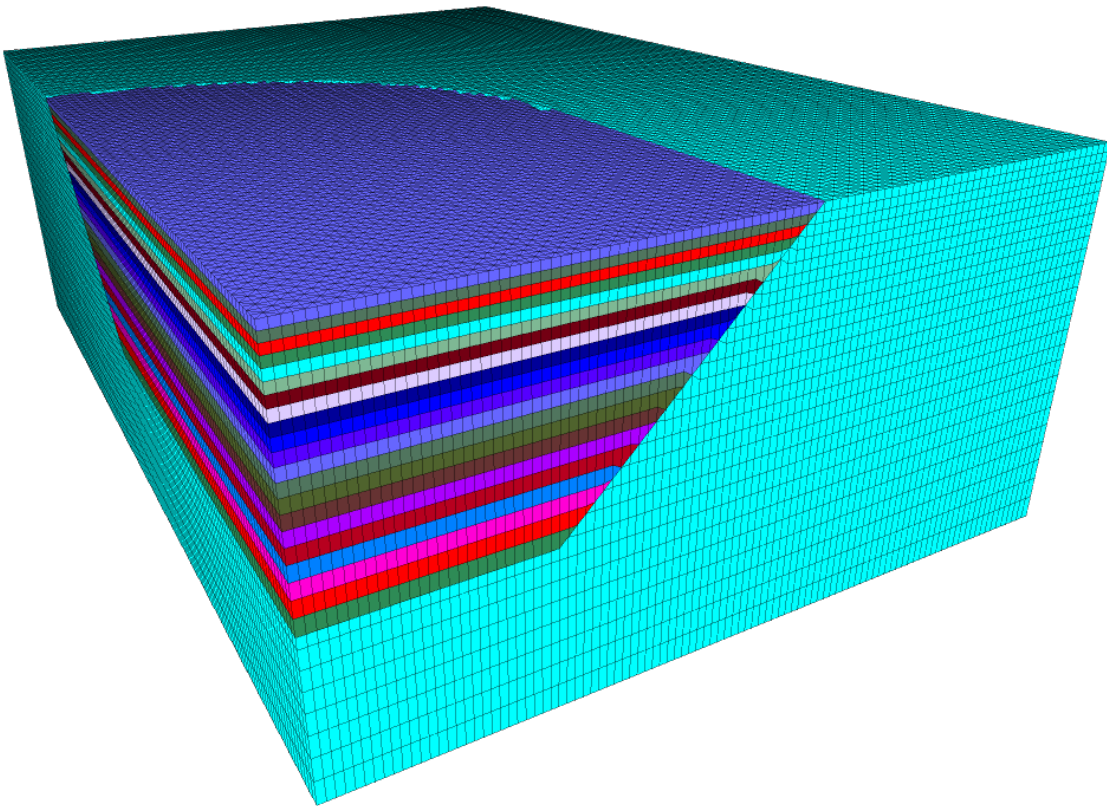
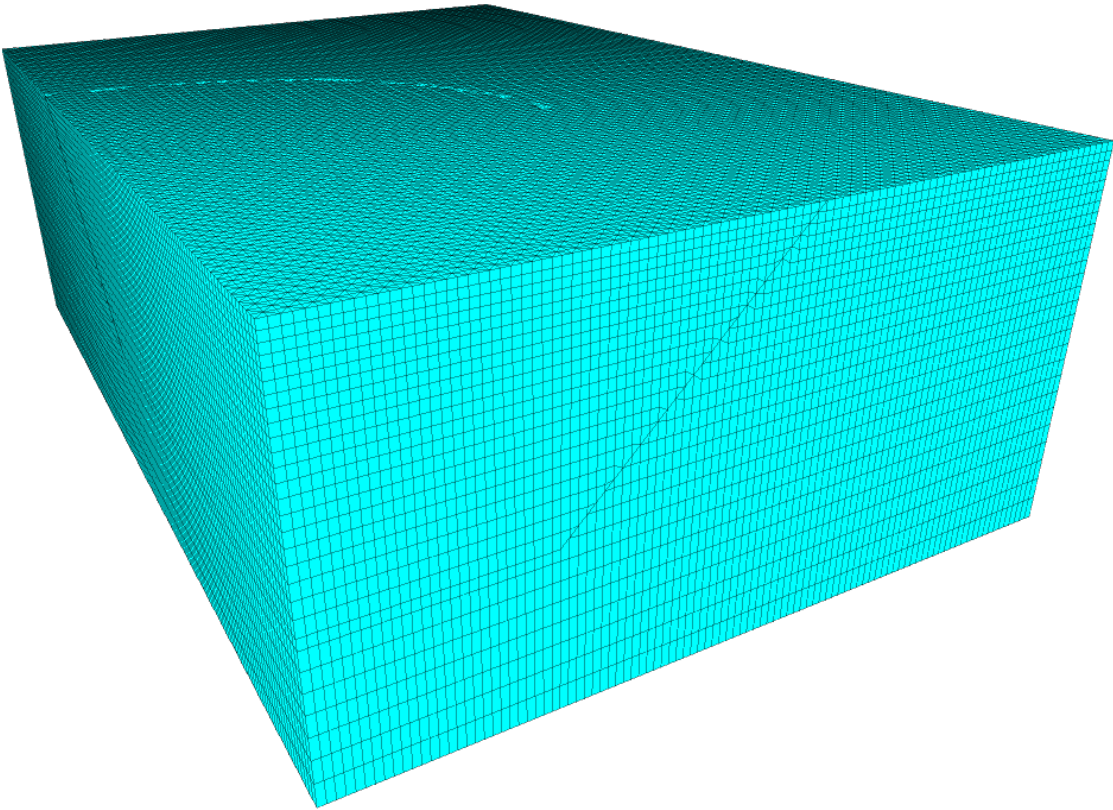


Figure 69: *Bathtub* full mesh. Upper image, full mesh in slot 1. Lower image, full mesh in slot 2 – image from FLAC3D©.

The upper image of [Figure 69](#) shows the full mesh plotted in slot 1. Notice this slot was specified in [Figure 68](#), right underneath where *Output File Name* was specified. Similarly, the lower image of [Figure 69](#) shows the full mesh plotted in slot 2. This last slot was specified right next to the *Multi Group* option.

One of the strongest features of [Geo Meshing v4.3](#) is the automatic layering option, as shown in the lower image of [Figure 69](#). This process can be quite challenging in other meshing tools, especially when an element height gradient is specified, as it was done in this example. This grouping allows for a ready implementation in FLAC3D© or either an excavation or a back fill of the *bathtub*. It is worth mentioning that the user can also specify a constant element height, while still activating the *Multi Group* option.

The user is encouraged to try different options in both, the *3D polygonal excavation* and the *3D meshing* dialog windows. By trying different option combinations, that are just a click away, the user will discover the real potential and flexibility of [Geo Meshing v4.3](#).

9.4. More Examples

Please use the following links to access more examples of [Geo Meshing](#). Although they were made in an older version, they still can be developed in the current version.

1. Tutorial 1 – Basic Concepts
https://youtu.be/A_6e-soKkn4
2. Tutorial 2 – Creating Topography from CAD file
<https://youtu.be/VeIFHclbyVI>
DXF file download
<https://www.geomeshing.com/wp-content/uploads/2017/08/topo-1.zip>
3. Tutorial 3 – Creating Topography from online Bing Map Service
<https://youtu.be/hln210Lae-c>
4. Tutorial 4 – Creating a Dam and filling its impoundment
<https://youtu.be/G1omcLjsHrM>
DXF file download
<http://www.geomeshing.com/wp-content/uploads/2017/08/topo-1.zip>