Geo Meshing

Geo Meshing v.6:

User Guide Mesh Generation & Tools

New User Interface. Enhanced Compatibility With Other Platforms

Email: geomeshing@gmail.com & help.geomeshing@gmail.com

November 2025

Contents

List of Figures

1. INTRODUCTION

Geo Meshing v6 is a software specifically designed for developing 3D meshes for FLAC3D®, 2D meshes for FLAC2D® and general purpose meshes using just tetrahedrons. Good examples of this last use of *Geo Meshing v6*, are as pre-processor for Material Point Methodology (MPM) analysis and as pre-processor for NUMGEO.

Geo Meshing v6 greatly simplifies the complex process of producing advanced 3D meshes for FLAC3D®, and therefore, reduces the cost associated with those tasks. Likewise, *Geo Meshing v6* effortless produces meshes for plain strain analysis, it takes a simple CAD type file (DXF) and quickly converts it to either FLAC3D® or FLAC2D® meshes for plain strain analysis.

Although *Geo Meshing v6* was specifically tailored for FLAC3D®/ FLAC2D®, it can also be used with other software by activating the "only tetrahedron" option which produces meshes compatible with a wide variety of software.

2. SERVICES OFFERED

In addition to the mesh generation capabilities of *Geo Meshing v6*, *Geo Meshing Engineering PTY LTD* provides a wide range of consulting services in both geotechnical engineering and/or numerical modeling. These services are delivered by professionals with over two decades of international experience.

2.1. **Geotechnical Services** (25 years of experience)

- Geotechnical expertise and advice
- Geotechnical report reviews
- Site inspection visits
- Coordination and planning of site investigation campaigns
- Laboratory testing program reviews
- Dam Safety Review (DSR)

2.2. **Numerical Modeling Services** (20+ years of experience)

- Mentoring in FLAC2D and FLAC3D modeling
- Material properties calibration based on laboratory and in-situ data
- Static and dynamic analysis in 2D and 3D
- MPM (Material Point Method) analysis
- Custom mesh generation for FLAC and other platforms
- Review of numerical models and interpretation of results
- Partnership support for numerical modeling projects

If you are interested in contracting the services of the independent consulting company *Geo Meshing Engineering PTY LTD*, please refer to the contact information provided in **Section 11** of this manual.

3. MAIN FEATURES

3.1. 3D Features

- **Layered mesh**: *Geo Meshing v6* 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 v6* creates, as much as possible, balanced meshes, where large contrasts in element sizes and orientations are avoided.

- **Parametric body definition**: *Geo Meshing v6* uses a parametric definition for creating a wide variety of bodies without the need of a CAD software.

- **CAD compatibility**: *Geo Meshing v6* can extract polyline information from DXF files. Using this feature, a topographic landform can be developed in minutes.

- **Bing Maps® Ready:** *Geo Meshing v6* can extract topographic surfaces directly from Bing Maps®, thus landforms can be easily imported into *Geo Meshing v6.*

- **GID® compatibility**: *Geo Meshing v6* reads and imports 2D files created with GID®. These files can be used to produce 3D meshes.

- **Meshing refinement**: *Geo Meshing v6* can refine meshes at any level within the model.

- **Interpolation algorithms**: *Geo Meshing v6* includes two interpolation algorithms, inverse distance squared and Kriging for obtaining smooth surface profiles.

- **Flexible**: *Geo Meshing v6* output files can be used with other software by means of the "only tetrahedron" option.

- **Multiprocessor**: *Geo Meshing v6* is a parallel processing software, allowing for faster development of meshes.

- **Export to FLAC3D® Geometry:** Any surface developed with *Geo Meshing v6* can be transformed and imported into FLAC3D® as a 3D geometry.

- **Compatible:** *Geo Meshing v6* is capable of producing meshes compatible with current and previous FLAC3D® versions.

- **Repeatability:** As a complex meshing process is developed with *Geo Meshing v6*, all steps are recorded. If at a later stage, a parameter is decided to be changed for the entire modelling process, the history file can be call and run all steps with the new parameters, without the need of running manually each step.

3.2. 2D Plane Strain Features

- **Layered mesh**: *Geo Meshing v6* 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 v6* creates, as much as possible, balanced meshes, where large contrasts in element sizes and orientations are avoided.

- **CAD compatibility:** *Geo Meshing v6* can extract polyline information from DXF files.

- **Easy to edit:** *Geo Meshing v6* generates a project history, that can be edited in *Geo Meshing v6*. Using the project history feature allows to further process the mesh and tailored it to a specific analysis.

- **Multiprocessor**: *Geo Meshing v6* is a parallel processing software, allowing for faster development of meshes.

- **Export to FLAC2D®/FLAC3D® Geometry:** Any surface developed with *Geo Meshing v6* can be transformed and imported into FLAC3D® as a 3D geometry or into FLAC2D® as a 2D geometry.

4. GEO MESHING V6 SETUP

Geo Meshing v6 can be installed automatically or manually. It is highly recommended to install it automatically.

Once *Geo Meshing v6* is installed in your computer, the files described in section 4.2 will show at the installation folder. In addition, a file called "Geo_meshing.key" also needs to be stored in this folder. See section 4.3 for instructions.

4.1. 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 shown in Figure 1. Follow the screen instructions to install *Geo Meshing v6*.

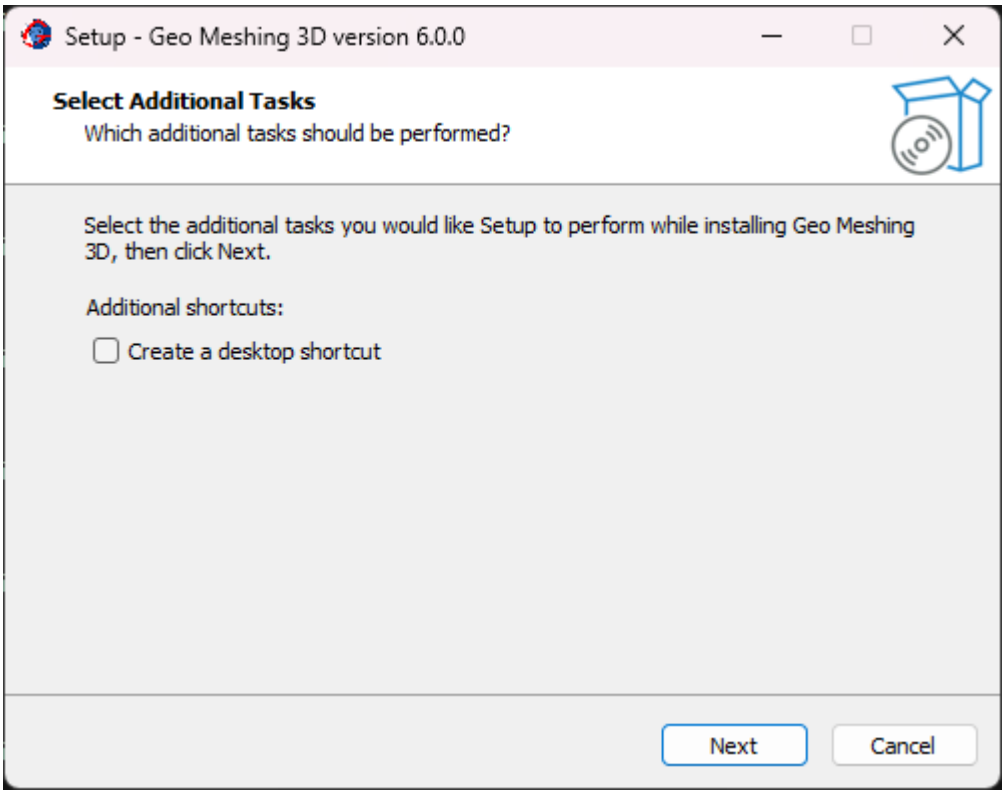


Figure 1: *Geo Meshing v6* Installation Dialog Window.

4.2. 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 v6 will not run properly if copied to a different source folder.*

This folder should contain the following files:

- *Geo Meshing v6.x.exe*
- *topo_render_v6.dll*
- *CPTu_plot_v3.dll*
- *bmaplib.dll*
- *DXF_Sketcker_v2.dll*
- *Microsoft.Maps.MapControl.WPF.dll*
- *log_change.txt*
- *GMManual6.pdf*
- *Resources [folder]*
- *MapData [folder]*
- *Examples [folder]*

4.3. Registration

Geo Meshing v6 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 a license.

You will also need to include in your email the *Geo Meshing v6* code. The *Geo Meshing v6* code can be obtained under *Help/About Geo Meshing* menu, AFTER the program has been installed (see Figure 2).

To avoid mistyped information, highlight *Geo Meshing v6* 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 v6* to activate it.

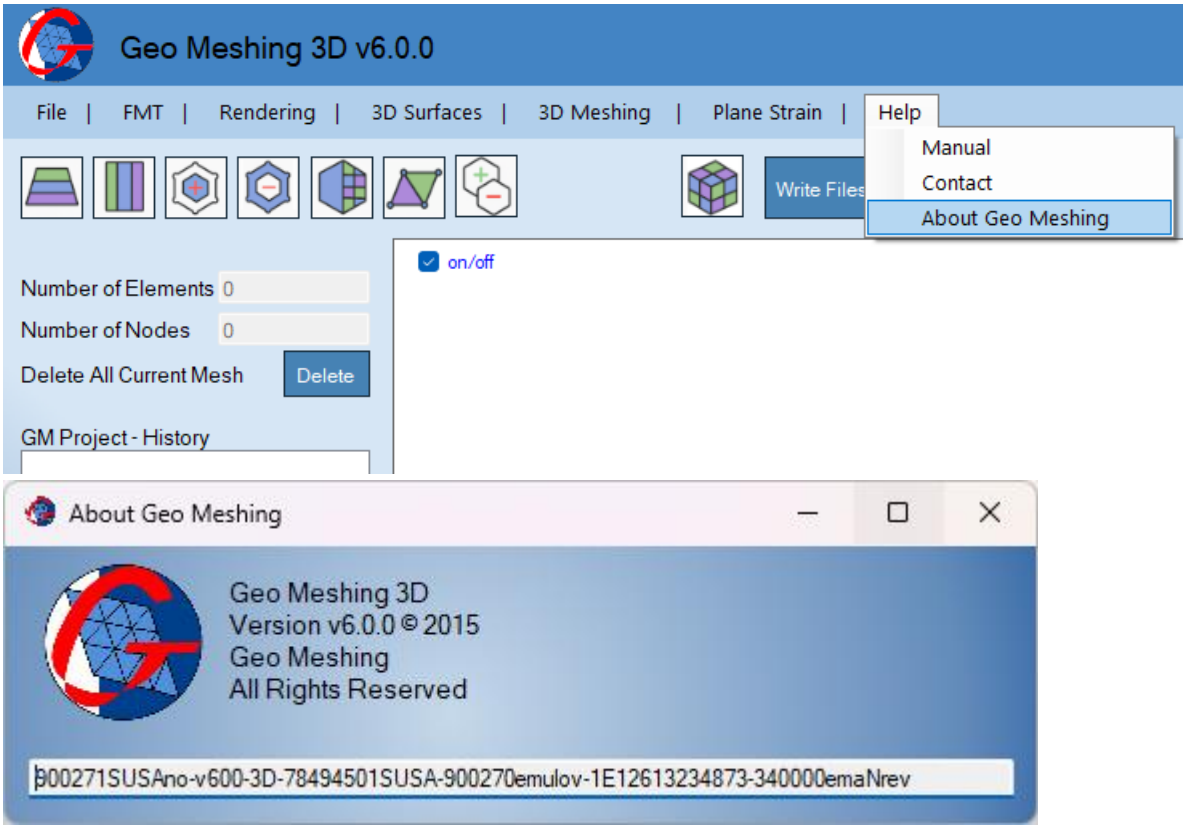


Figure 2: *Geo Meshing v6* Code.

5. GEO MESHING V6 OVERVIEW

Figure 3 shows the main components of *Geo Meshing v6*.

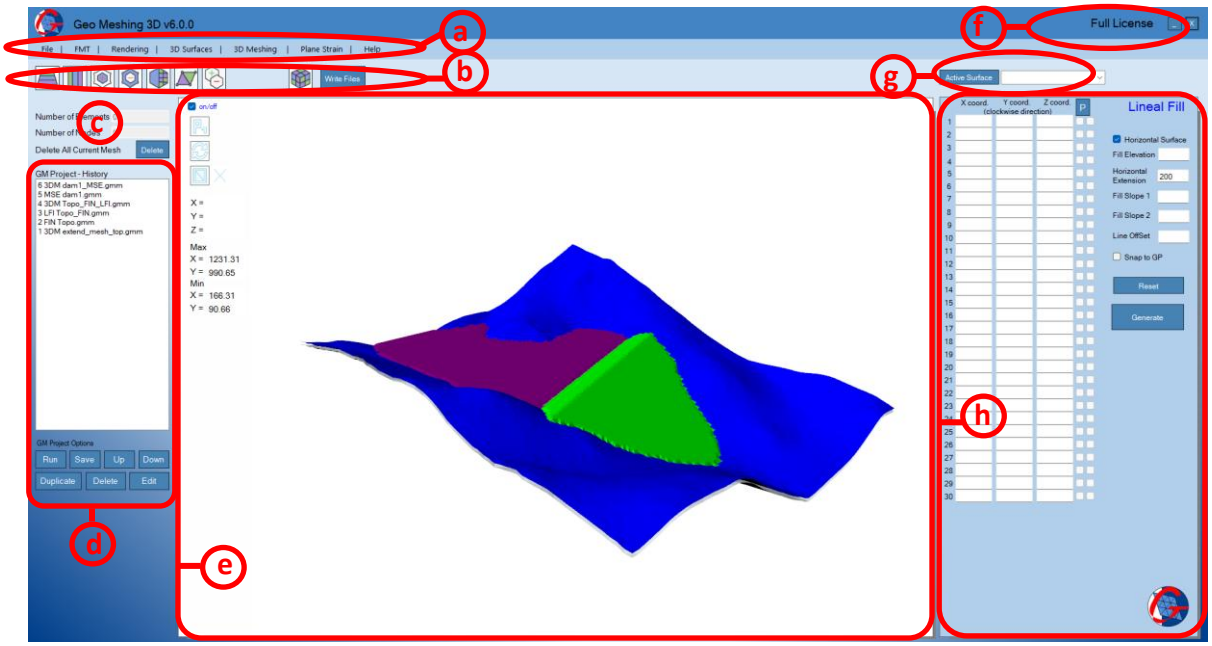


Figure 3: *Geo Meshing v6* Overview.

a) Main Menu: The strip menu area provides options for *Geo Meshing v6.* 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) Tool Bar: 3D Surfaces, 3D Meshing and Write Files: It provides easy access to all *3D Surfaces* tools. Also, it provides access to the *3D meshing* tool and *Write Files*. This last button allows to write mesh and Master Surface files.

c) This area displays the total number of elements and nodes generated by the commands recorded in the *GM Project – History* window.

d) GM Project – History: This area tracks history of the meshing events (see section 5.1 and 6.1.1).

e) **Render**: This window displays information for 2D and 3D options. When using 3D options, the last surface/mesh generated with *Geo Meshing v6* is shown. This procedure is called after the *3D Meshing* algorithm is executed (see section 6.5). When using the 2D options, this window displays the 2D imported geometry. Finally, *Geo Meshing v6* can show surface/mesh or 2D geometry when *Rendering* option is used (see section 6.3)

f) **License type:** This area displays the type of license under use in *Geo Meshing v6*, it can be "No License", "Test License" or "Full License" (as shown).
Note: Test license is often referred as Free Version

g) **Active Surface:** This area displays the *Active Surface* on which *Geo Meshing v6* tool will develop the 3D surface.

h) **3D Surface Tool:** This area shows the current 3D surface tool.

5.1. GM (Geo Meshing) Project Options

- Geo Meshing Project Options has seven options, which are shown in Figure 4.

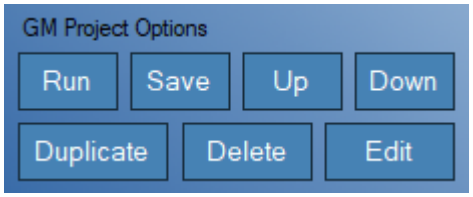


Figure 4: *Project Options*.

  - **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 v6* Project – History windows. Before using this option, select an event in the history window.

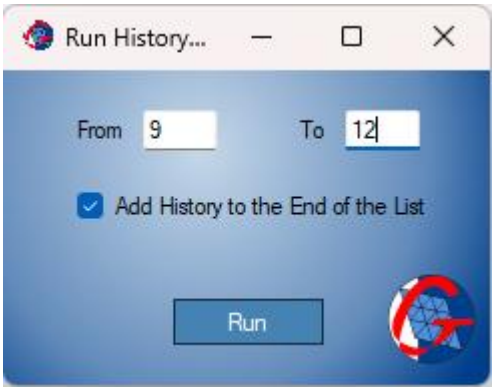


Figure 5: Geo Meshing History – Run, define range dialog window.

- **Save:** This option saves the entire *Geo Meshing v6* history to the hard drive. If a file name for the *Geo Meshing v6* history has not been defined, a window will prompt requiring a name during the saving process.
- **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.
- **Duplicate:** This option duplicated the selected event.
- **Delete:** This option deletes the event selected in the *Geo Meshing v6* history window. Before using this option, select an event in the history window.
- **Edit:** This option enables to edit the recorded events in the history window (section 5d)). Figure 6 shows the dialog windows for this option. Currently, only *"3D Meshing"* and *Finner* events are editable.

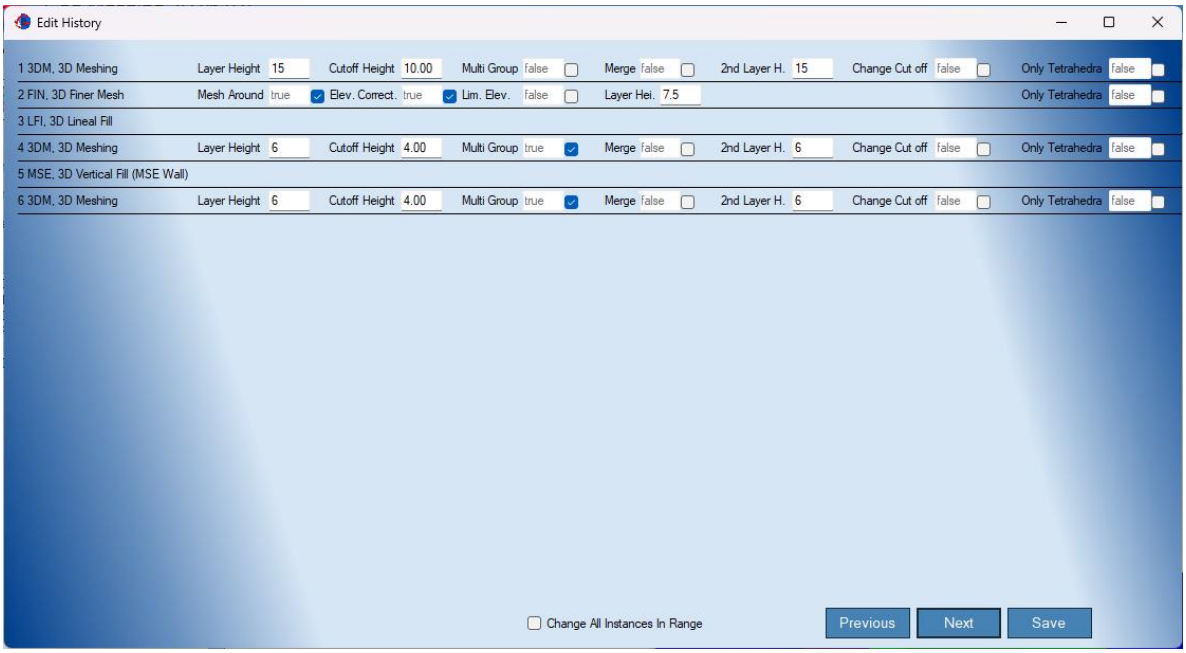


Figure 6: Geo Meshing History – Edit, edits option in history.

6. MAIN MENU

6.1. File Menu

6.1.1. Open/Create Project

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

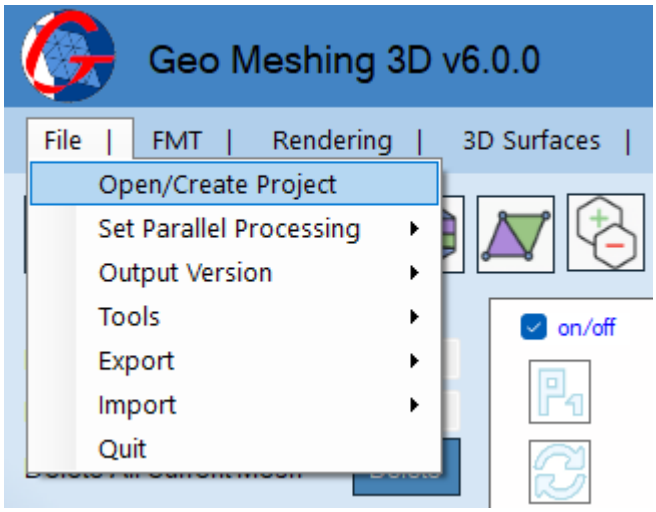


Figure 7: File Menu – Open/Create Project.

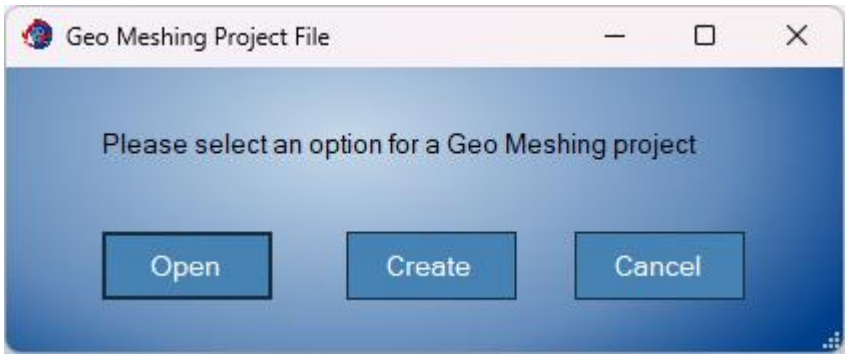


Figure 8: Open/Create Project dialog window.

6.1.2. Set Number of Processor Menu

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

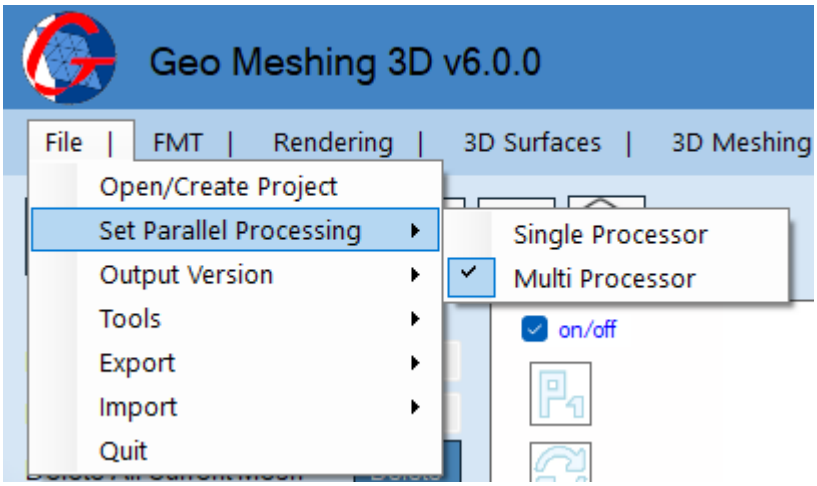


Figure 9: File Menu – Set Number of Processors.

- **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 v6* uses the maximum number of threads available.

  - **Hint:** The larger the number of processors the faster the meshing process.
  - **WARNING:** Proper ventilation must be provided to the PC/WS/Notebook/Laptop, otherwise it may overheat.

6.1.3. Output Version

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

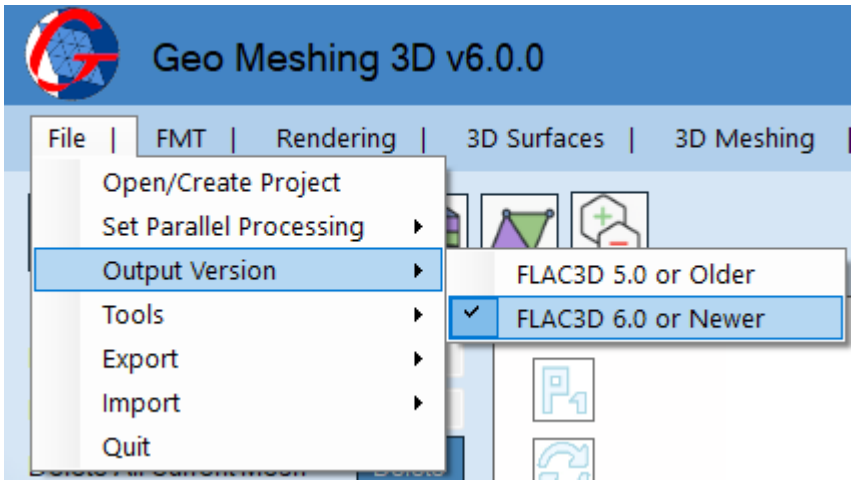


Figure 10: File Menu – Output Version.

- **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.

6.1.4. Tools

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

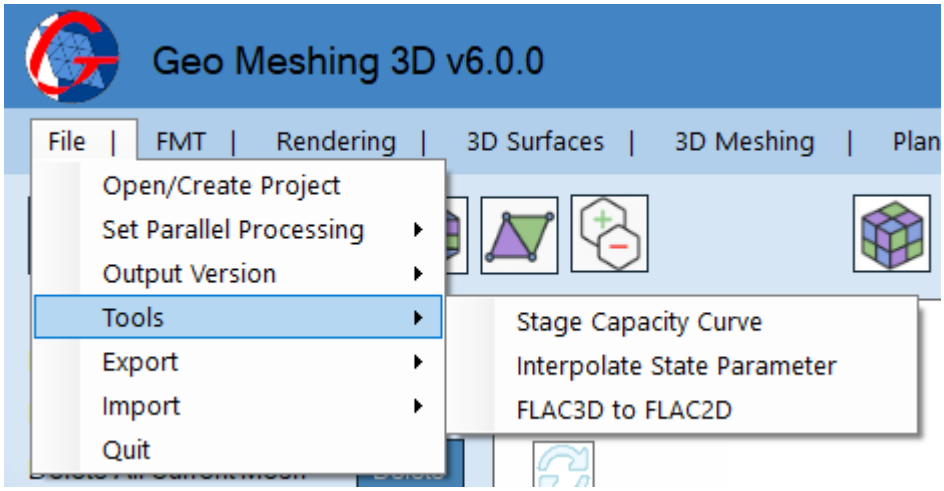


Figure 11: File Menu – Tools.

- **Stage Capacity Curve:** Opens the dialog window shown in Figure 12. This feature computes the volume versus elevation between two 3D surfaces (section 6.4) and records the data in a text file.
  - **Open Top Surface:** Opens a *Geo Meshing v6* file, with the 3D surface containing the higher elevation.
  - **Output Bot. Surface:** Opens a *Geo Meshing v6* 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.

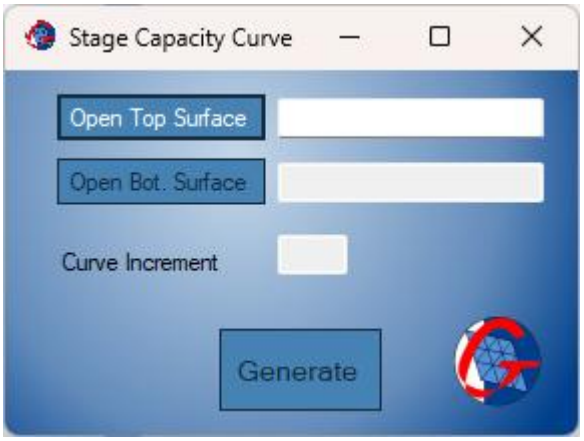


Figure 12: Stage Capacity Curve dialog window.

- **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.

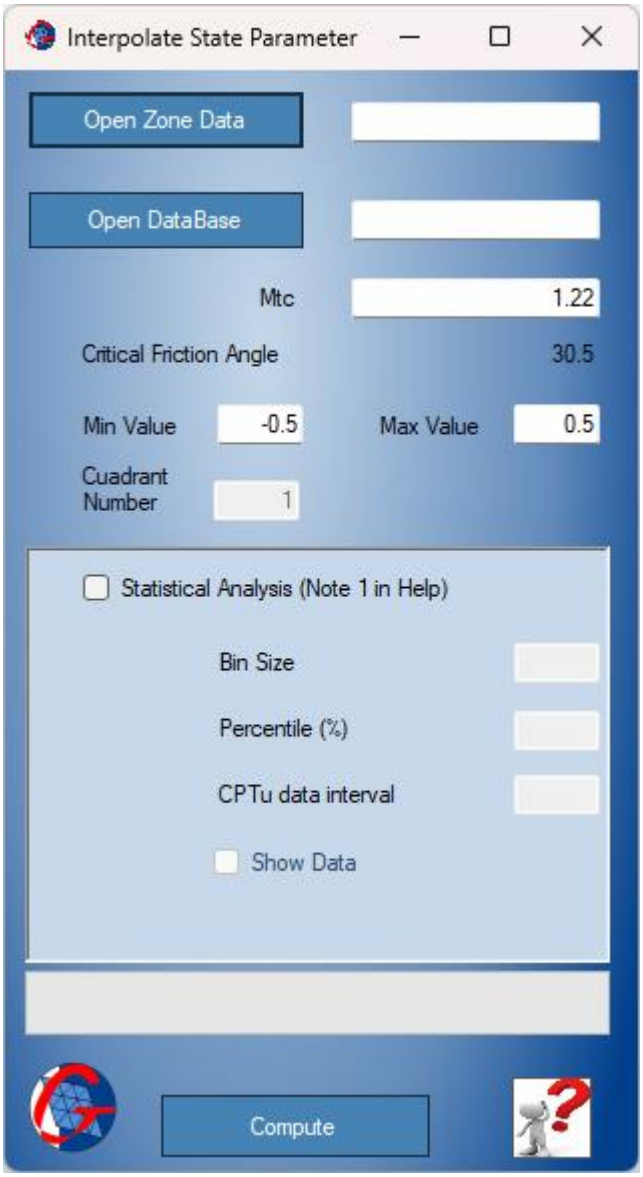


Figure 13: Interpolate State Parameter dialog window.

- **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 thana 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.

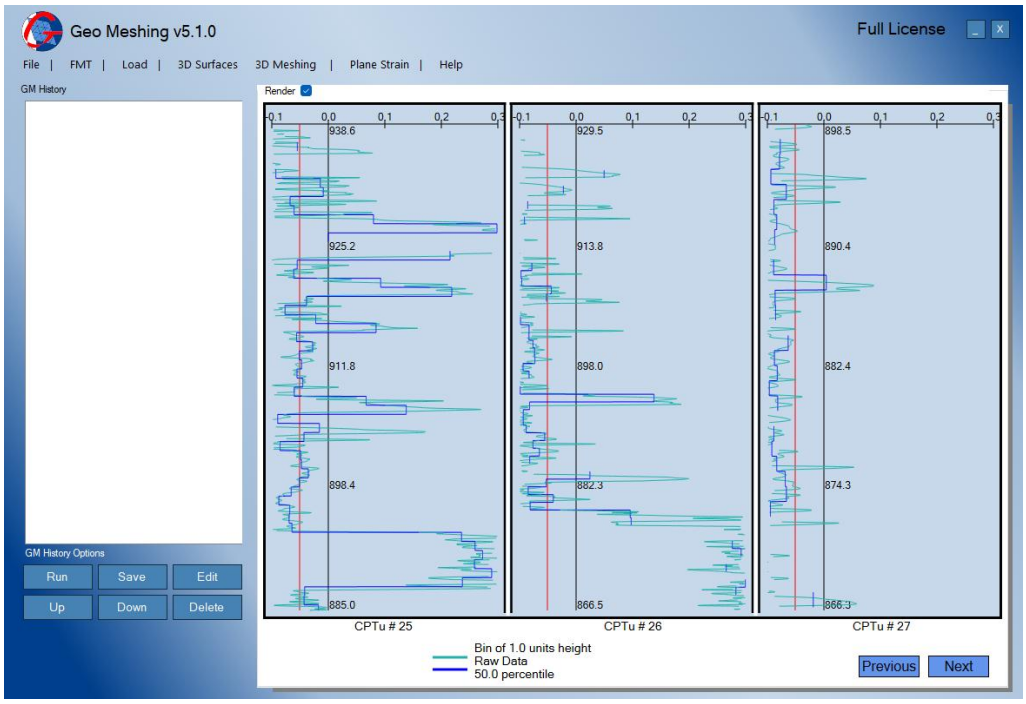


Figure 14: State Parameter Statistical Analysis.

- **FLAC3D to FLAC2D:** Opens the dialog window shown in Figure 15. This option transforms existing FLAC3D® meshes to FLAC2D meshes. The process is done for a specific Y-plane only. The Y-plane must coincide with the nodes coordinates to the second decimal point.
  - **FLAC3D Gris:** Opens a file with the FLAC3D mesh information.
  - **Y Plane Coordinate:** Defines the Y-Plane coordinate at which the 2D model will be created. The Y-Plane coordinate must match the coordinates of the gridpoints forming that plane.
  - **Generate File:** Starts the process.

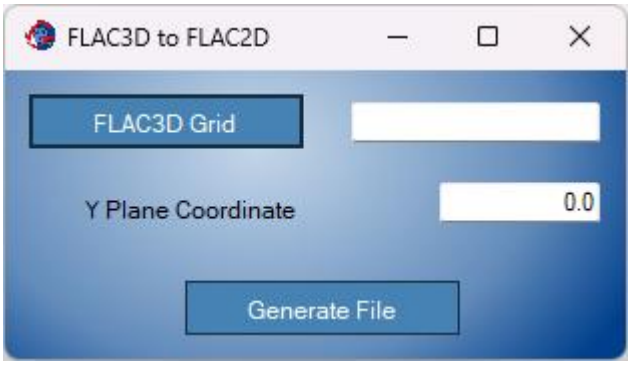


Figure 15: FLAC3D to FLAC2D dialog window.

6.1.5. 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*.

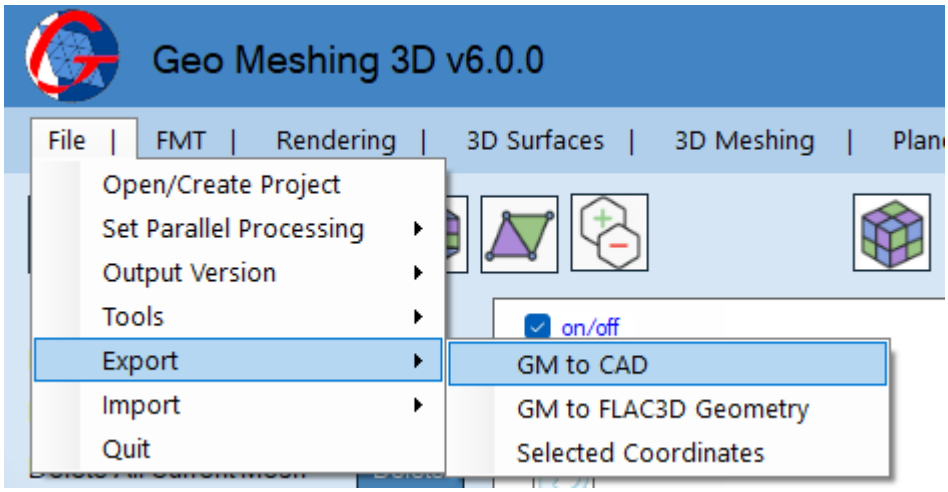


Figure 16: File Menu – Export.

  - **GM to CAD Script:** Opens a dialog window (shown in Figure 17) for converting *Geo Meshing v6* surfaces 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 v6* 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).

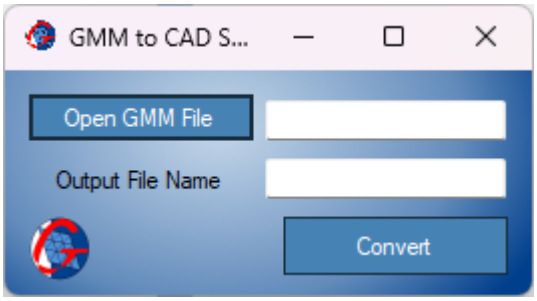


Figure 17: *Geo Meshing v6* to CAD script surface dialog window.

- **GM to Flac3D® Geometry:** Opens a dialog window (shown in Figure 18) for converting *Geo Meshing v6* 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 v6* 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.

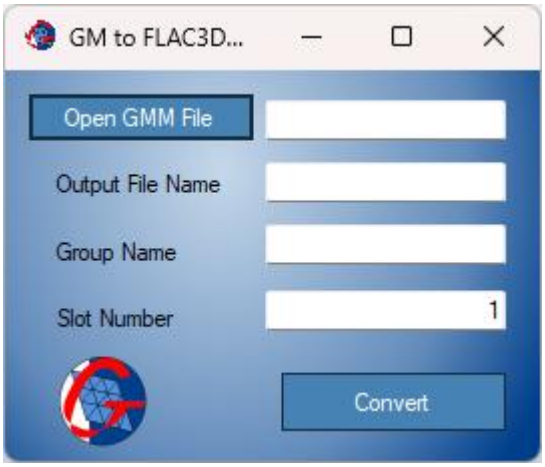


Figure 18: *Geo Meshing v6* to Flac3D® Geometry dialog window.

- **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 7.2).
  - **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 7.2).
  - **Reset:** Deletes the current coordinates.
  - **Generate:** Export xyz coordinates.

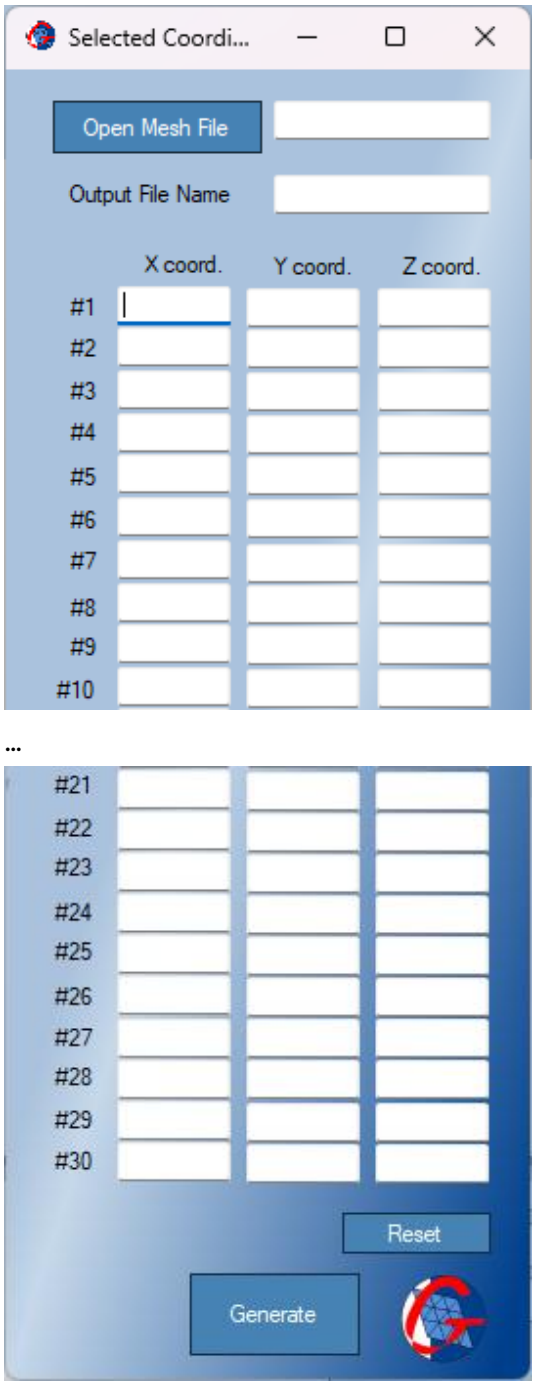


Figure 19: File Menu – Selected Coordinates dialog window.

6.1.6. Import

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

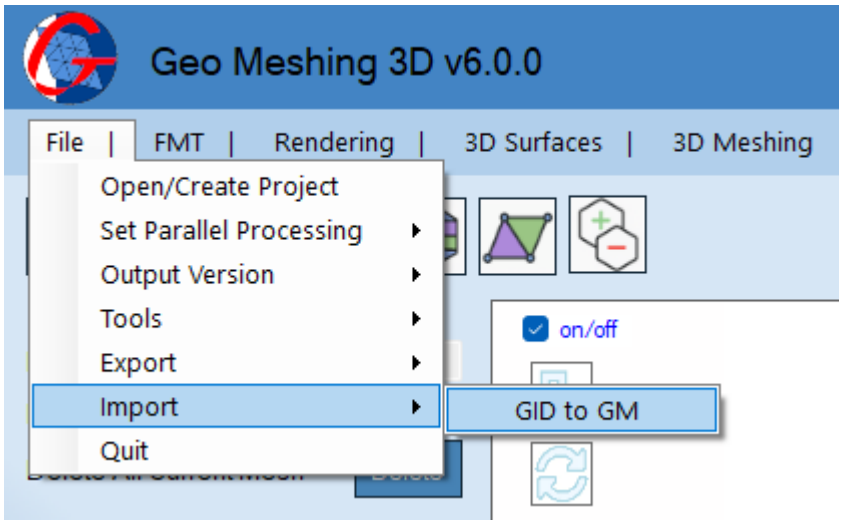


Figure 20: File Menu – Import.

  - **GID® to GM:** Opens a dialog window (shown in Figure 21) for converting GID® meshes files (*.msh) to *Geo Meshing v6* 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 v6* Mesh).
    - **Convert:** Starts the process.

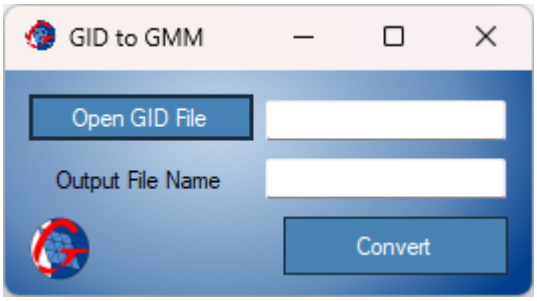


Figure 21: GID® to *Geo Meshing v6* dialog windows.

6.1.7. Quit

- **File Menu, Quit**: Exits from *Geo Meshing v6* (shown in Figure 22).

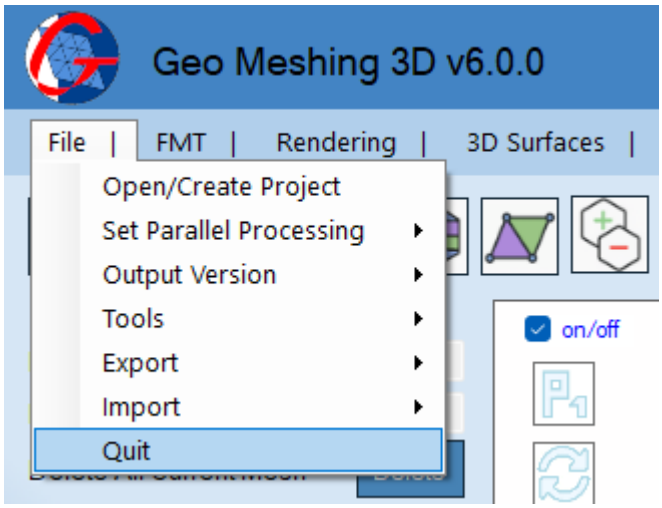


Figure 22: File Menu – Quit.

6.2. FMT (Flat Mesh & Topography)

This option opens the dialog window shown in Figure 23.

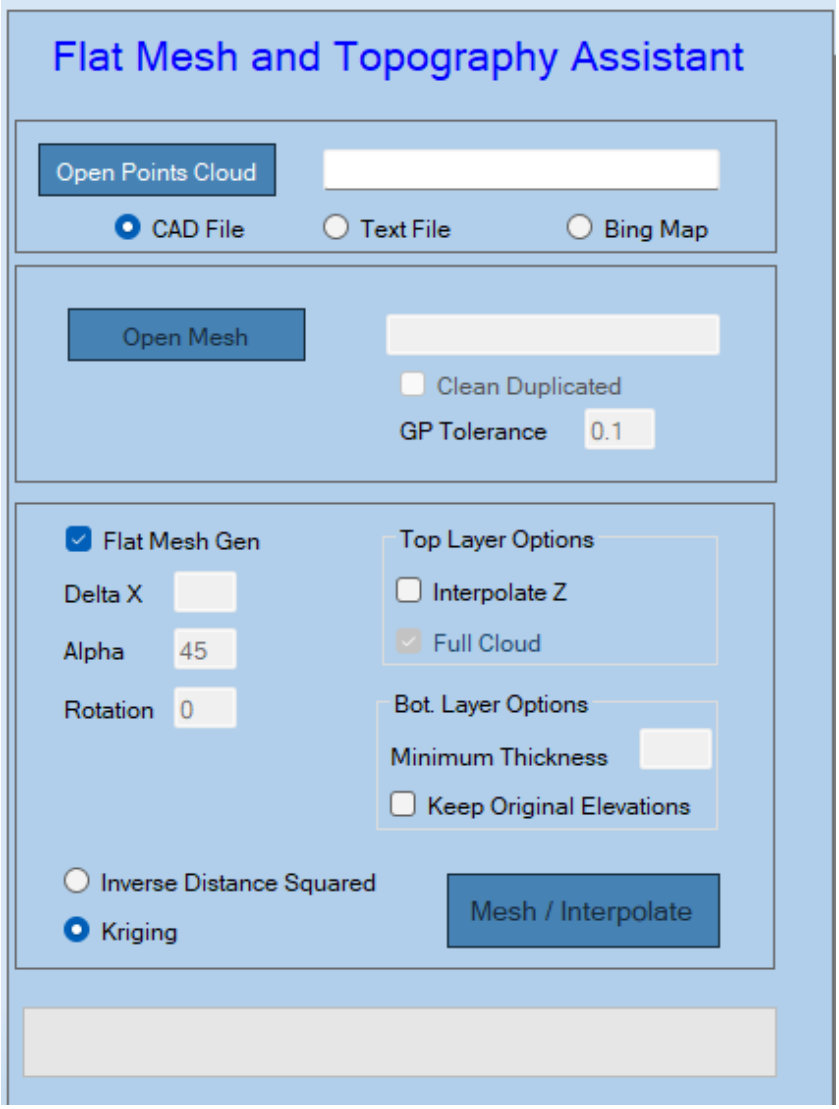


Figure 23: FMT (Flat Mesh & Topography) Menu.

- **Open Points Cloud**: This option helps selecting a cloud of point that represents a surface. Using this cloud of points a *Geo Meshing v6* 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 v6* 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 v6*, 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.

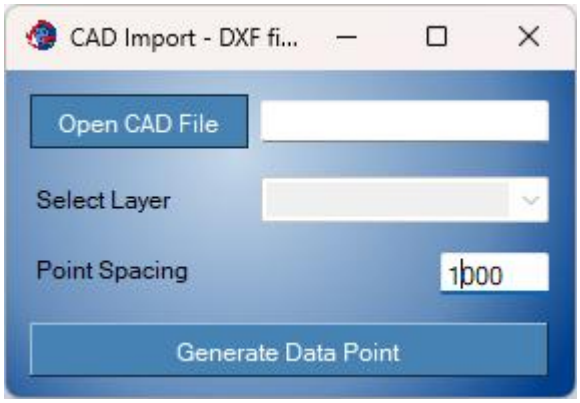


Figure 24: CAD Point Import.

- **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 v6* 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 you to select 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:

$X1_1\ Y1_1\ Z1_1$
$X2_1\ Y2_1\ Z2_1$ } Triangle 1
$X3_1\ Y3_1\ Z3_1$

.
.
.

$X1_n\ Y1_n\ Z1_n$
$X2_n\ Y2_n\ Z2_n$ } Triangle n
$X3_n\ Y3_n\ Z3_n$

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, 6.1.2). This option is useful when the topography is relatively 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.

6.3. Rendering

This option opens the dialog window shown in Figure 25.

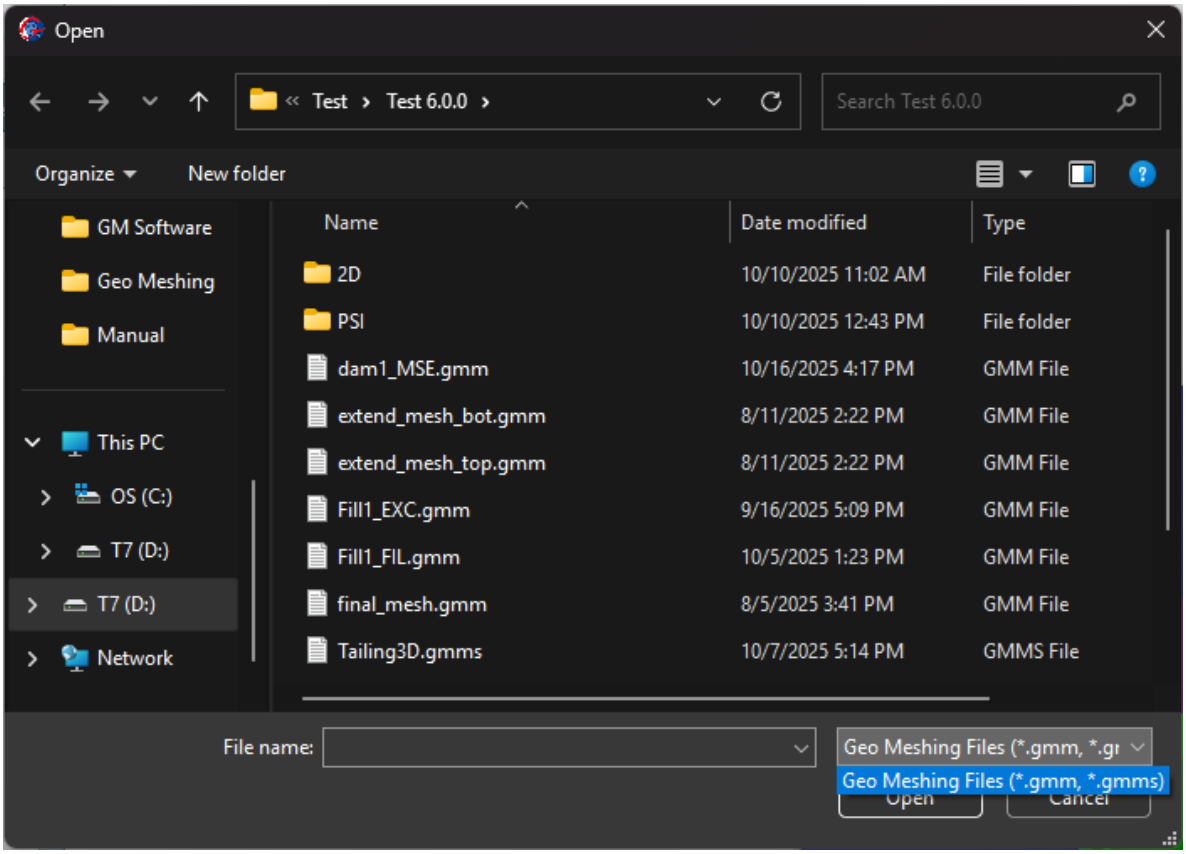


Figure 25: Rendering Open Dialog Window.

- Navigate to the folder where you have your *Geo Meshing v6* files stored.
- Select either *.gmm file (for *Geo Meshing* 3D surfaces) or *.gmms file (for *Geo Meshing* 3D Master Surface, Project).
- The selected surface or geometry is shown in the *render* area.

6.4. 3D Surfaces Menu

This option creates a landform over any given surface. Figure 26 shows the direct access icons for all *3D Surface* tools.

The landforms created by the *3D Surface* tools are defined parametrically, thus there is no need to develop complex CAD models to obtain a 100% compatible landform into a given 3D surface, for example a topography. Once the landform

is generated, the *3D Meshing* tool (section 6.5) 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.

- **Hint:** To visualize the surfaces or to export the mesh on the rendering window, either *3D Meshing* or *Finer Mesh* tools need to be executed.

Figure 26: 3D Surface Icons.

Figure 27 shows the *Active Surface* drop down menu. In order to use any of the tools described in this section, and active surface must be selected in forehands.

Figure 27: Active Surface Drop Down Menu.

6.4.1. Removed Features

3D Dam Surface, *3D Circular Excavation Surface*, 3D Horizontal Finer Mesh and 3D Wedge Excavation Surface are no longer available in *Geo Meshing v6*.

These features have been superseded as follows:

- The *3D Dam Surface* option has been replaced by Linear Fill, which is a much more flexible tool (see 6.4.2).
- The *3D Circular Excavation Surface* has been replaced by *3D Polygonal Excavation Surface*. The later has been enhanced to accomplish circular excavation among many other shapes (see 6.4.4).
- The *3D Horizontal Finer Mesh Surface* has been replaced by *3D Finer Mesh Surface* (see 6.4.6).
- 3D Wedge Excavation Surface has also been replaced by *3D Polygonal Excavation Surface* (see 6.4.4).

6.4.2. 3D Linear Fill Surface (not active with *Test License*)

- **3D Surfaces Menu, Linear Fill:** This option (shown in Figure 28) defines a Linear Fill landform over any given surface (see 3D example in Section 10.2.3).

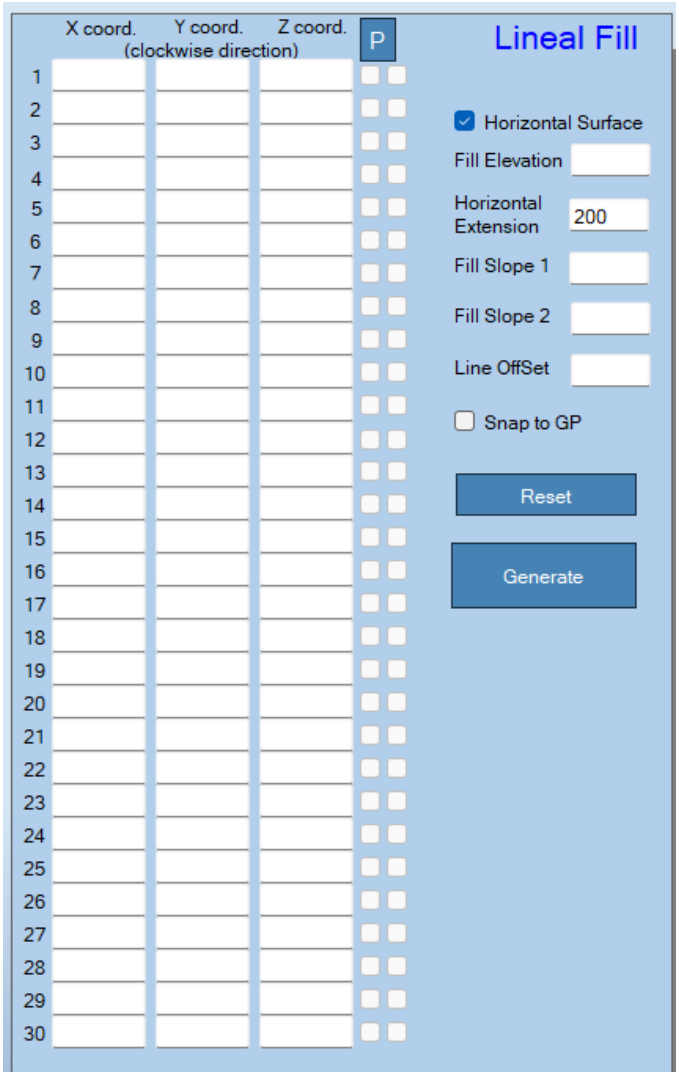


Figure 28: 3D Surfaces Menu – Linear Fill.

- **Open Mesh File**: Opens the *.gmm file containing the surface mesh, where the Lineal Fill will be created.

- **Xi, Yi and Zi Coordinates**: These coordinates define the plan view and the elevations of the linear fill. The linear fill coordinates support up to 30 vertices and the alignment should not cross over itself.
  - **Hint:** Coordinates for the Linear Fill can be selected using the mouse and the 3D render "F" projection (see section 7.3) or using the paste feature (below).
  - **"P":** will paste coordinates from a spreadsheet or a text file. Coordinates must be organized as follows:
    X1 Y1 Z1
    X2 Y2 Z2
    X3 Y3 Z3 …

- **Horizontal Surface:** When this option is selected, the linear fill will have a flat surface at the top area, with a constant elevation defined by *Fill Elevation*. Otherwise, *Geo Meshing v6* 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**: Linear Fill elevation at top surface.

- **Horizontal Extension**: Defines the extension of the linear fill beyond the area defined by the crest polygon.

- **Fill Slope 1**: Slope for the Linear 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.
- **Fill Slope 2:** As Fill Slope 1.

- **Line Offset:** Defines the width of the crest polygon.

- **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 Linear Fill surface process.

6.4.3. 3D Vertical Fill (MSE Wall) Surface 

- **3D Surfaces Menu, Vertical Fill (MSE Wall):** This option (shown in Figure 29) defines a Vertical Fill (MSE Wall) landform over any given surface.

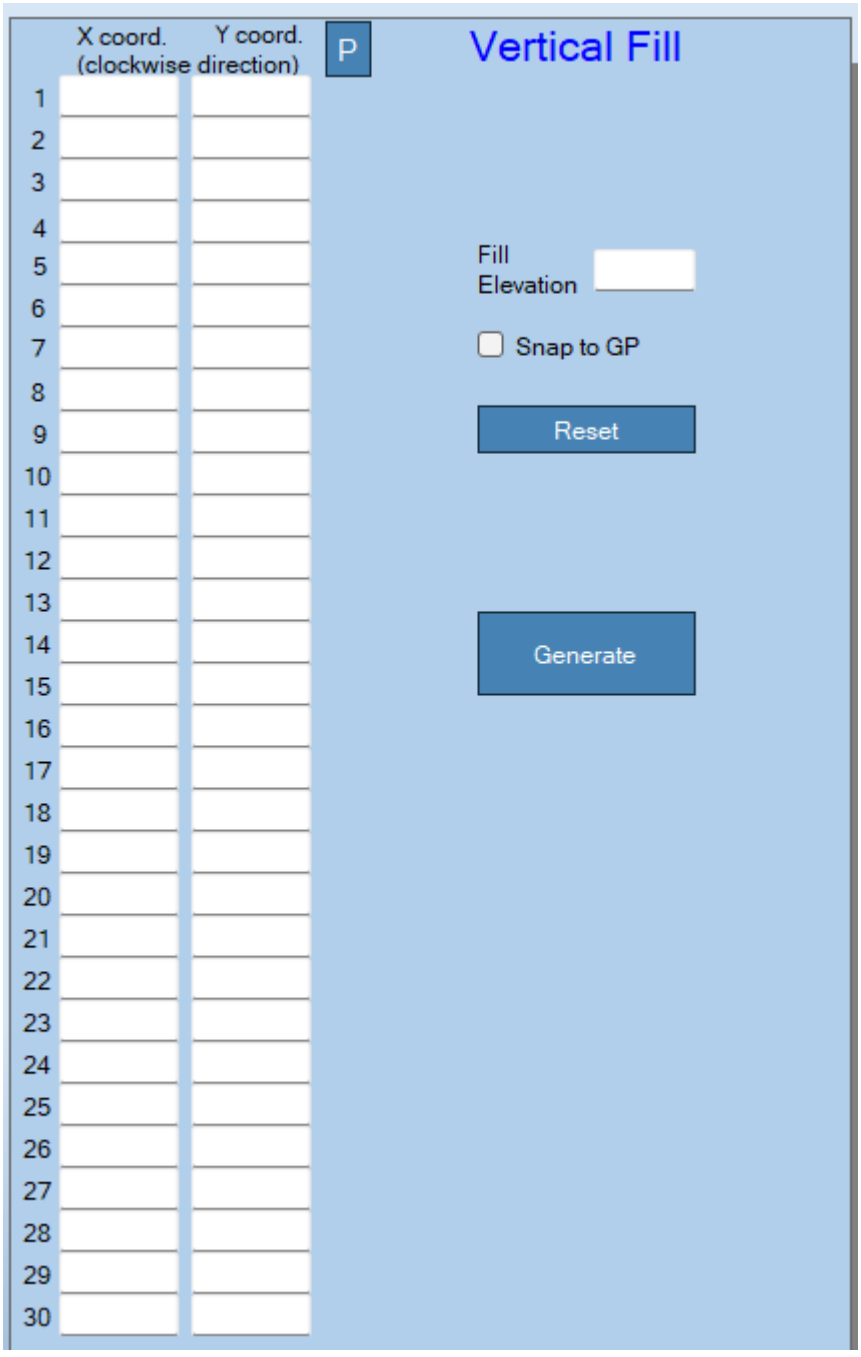


Figure 29: 3D Surfaces Menu – Vertical Fill (MSE Wall).

- **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 30 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 7.3) 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.

6.4.4. 3D Polygonal Fill Surface (not active with *Test License*)

- **3D Surfaces Menu, Polygonal Fill:** This option (shown in Figure 30) defines a Polygonal Fill landform over any given surface.

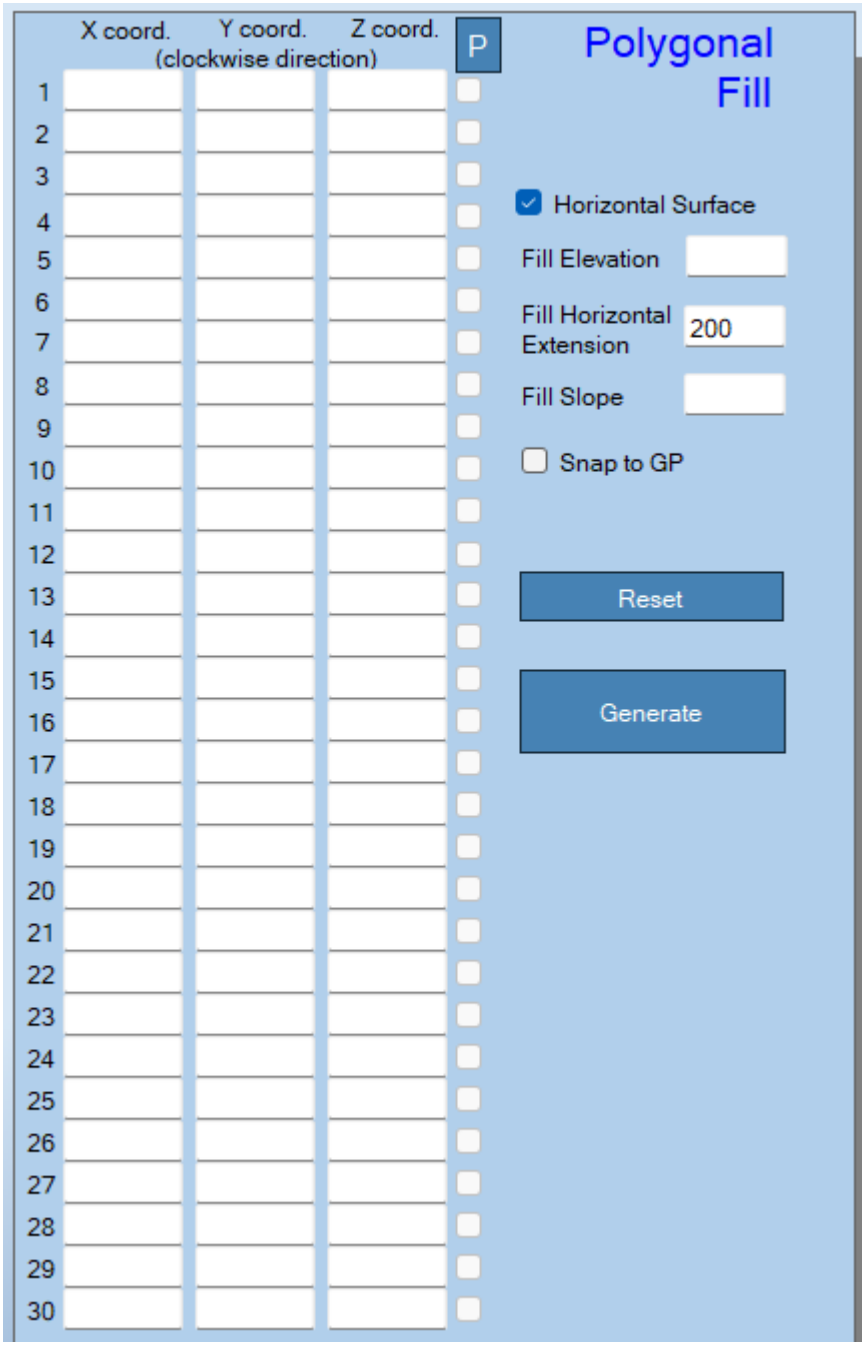


Figure 30: 3D Surfaces Menu – Polygonal Fill.

- **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 30 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 7.3) 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 v6* 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 Horizontal Extension**: Defines the extension of the fill beyond the area defined by the crest polygon.

- **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.

6.4.5. 3D Polygonal Excavation Surface (not active with *Test License*)

- **3D Surfaces Menu, Polygonal Excavation:** This option (shown in Figure 31) defines a Polygonal Excavation landform over any given surface.

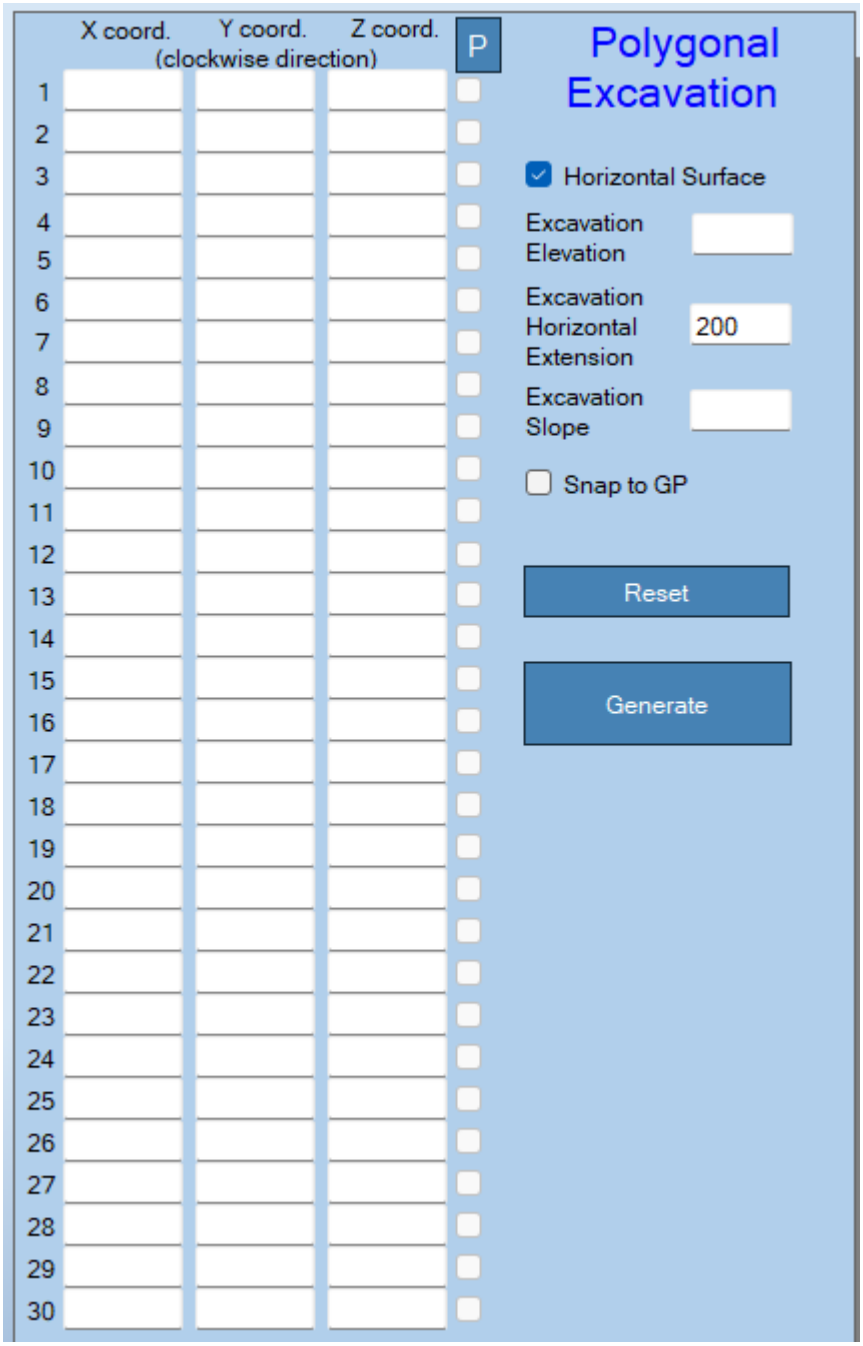


Figure 31: 3D Surfaces Menu – Polygonal Excavation.

- **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 30 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 7.3) 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 v6* 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 Horizontal Extension**: Defines the extension of the excavation beyond the area defined by the crest polygon.

- **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.

6.4.6. 3D Finer Mesh Surface (not active with *Test License*) 

- **3D Surfaces Menu, Finer Mesh:** This menu option (shown in Figure 32) 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*.

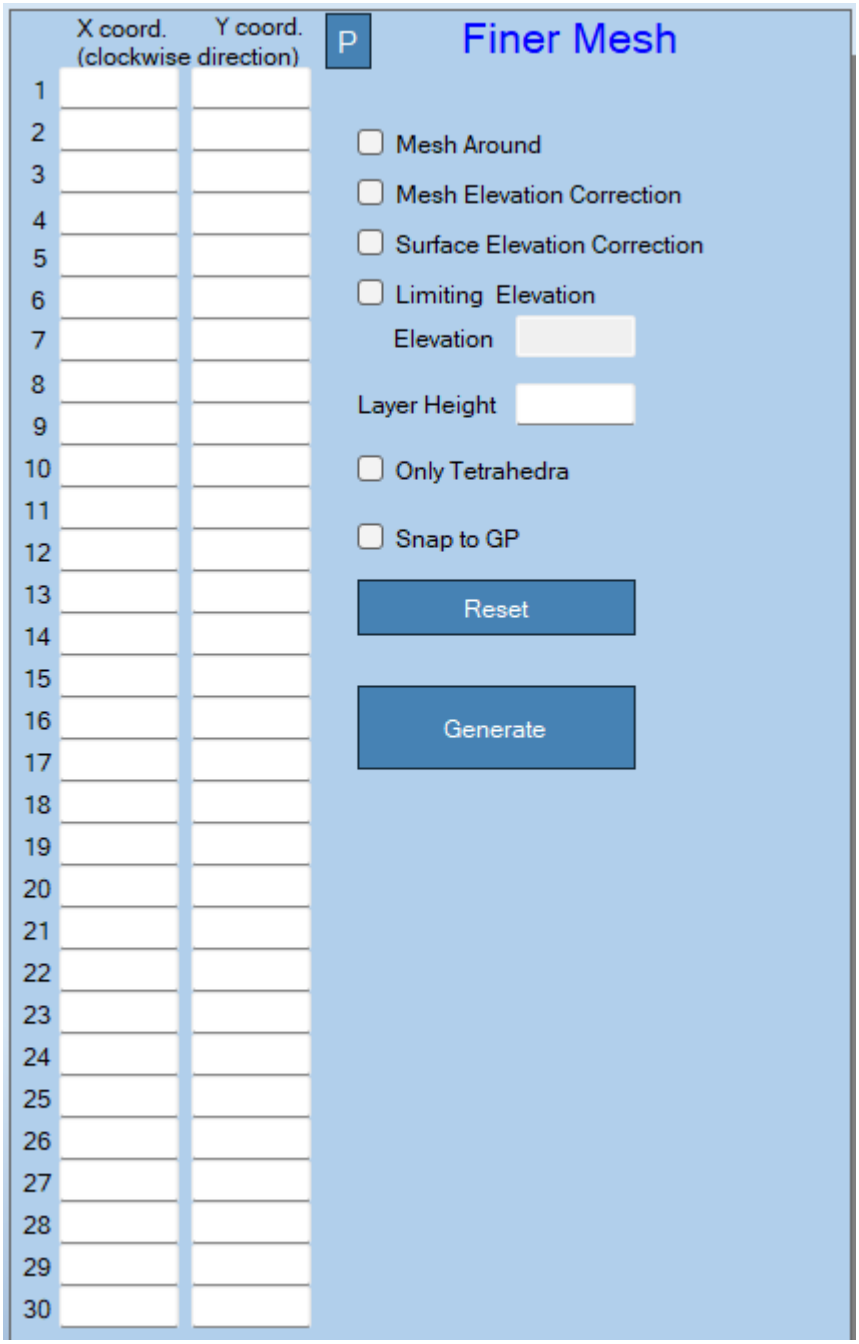


Figure 32: 3D Surfaces Menu – Finer Mesh.

- **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 30 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 7.3) 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 …

- **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.

- **Only Tetrahedra**: This option generates meshes using only tetrahedra. When this option is disable *Geo Meshing v6* uses the standard FLAC3D elements, i.e. bricks, wedges, pyramids and tetrahedrons. This option is useful for using the meshes in other programs rather than FLAC3D.

- **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.

6.4.7. 3D Interface Surface (not active with *Test License*)

- **3D Surfaces Menu, Interface Mesh:** This menu option (shown in Figure 33) produces a code for creating interfaces over an irregular surface created with *Geo Meshing v6*. This option does not require the execution of *3D Meshing* after generation.

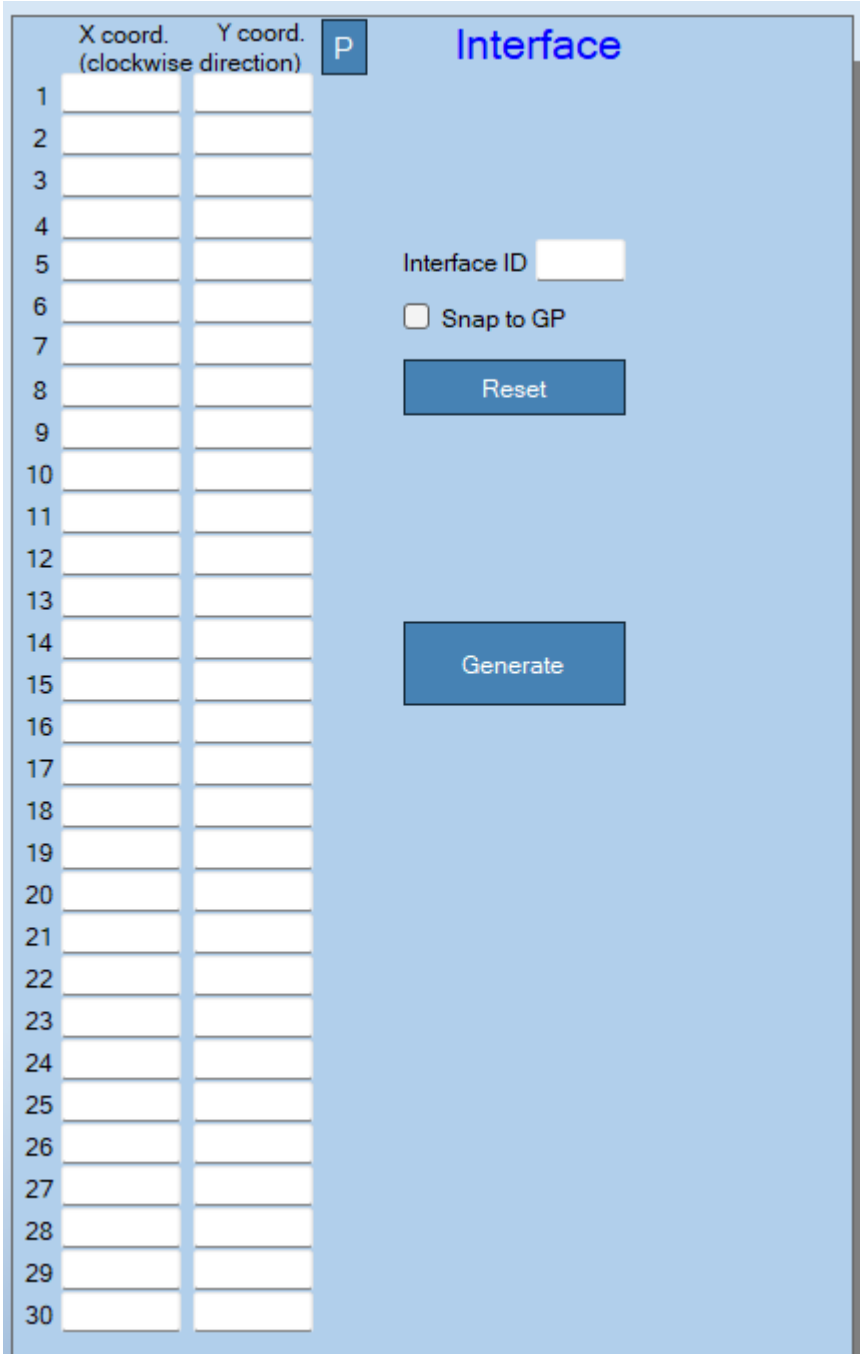


Figure 33: 3D Surfaces Menu – Interface Mesh.

- **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 30 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 7.3) 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.

6.4.8. 3D Add / Subtract Surface (not active with Test License)

- **3D Surfaces Menu, Add / Subtract Mesh:** This menu option (shown in Figure 34) operates an Addition or Subtraction between two 3D surfaces.

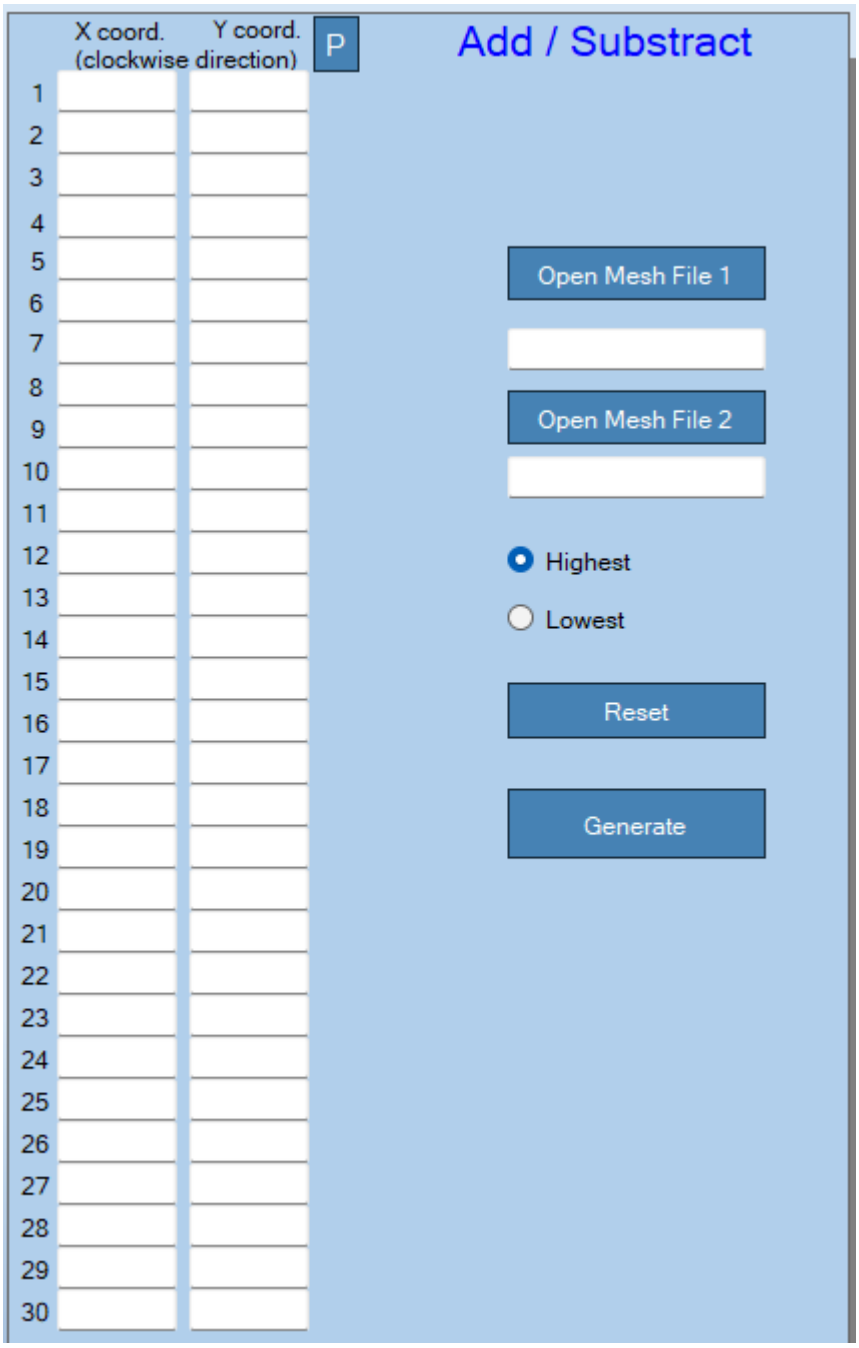


Figure 34: 3D Surfaces Menu – Add / Substract.

- **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 30 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 7.3) 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.

6.5. 3D Meshing

This option executes *3D Meshing* option, as shown in Figure 35.

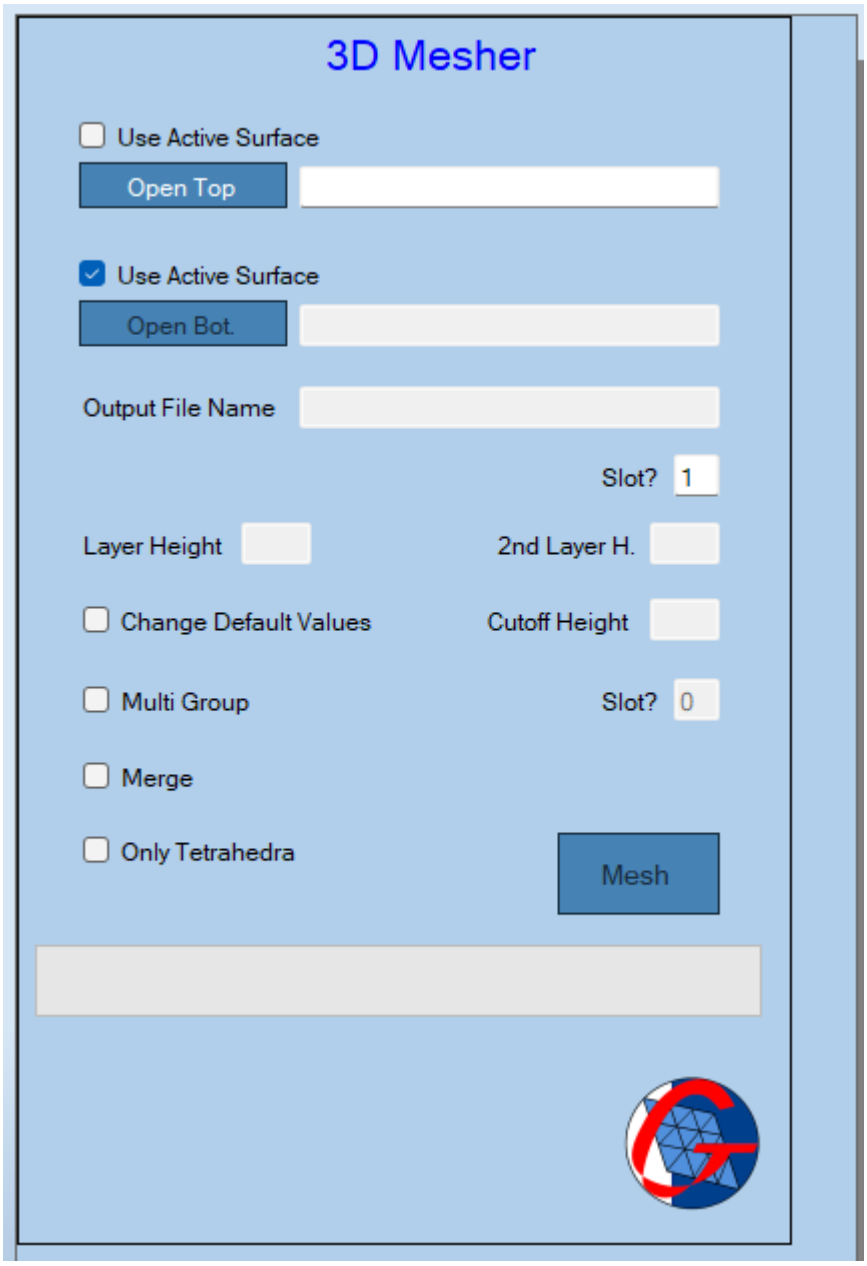


Figure 35: 3D Meshing Section.

- **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 6.4, the *Open*

Top file is automatically loaded, except when *Use Active Surface* is selected.

- **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 6.4, the *Open Bot.* file is automatically loaded, except when *Use Active Surface* is selected.

- **Use Active Surface**: This option selects the active surface for either the *Top* or *Bottom Surface*. The active surface is stored in memory, and therefore, there is no need to specify a file location. The active surface is usually generated after one of the tools from section 6.4 is successfully utilized.

- **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 disable *Geo Meshing v6* uses the standard FLAC3D elements, i.e. bricks, wedges, pyramids and tetrahedrons. This option is useful for using the meshes in other programs rather 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.

6.6. Plane Strain

This option creates meshes for Plane Strain analysis, either in FLAC3D or in FLAC2D. The dialog window is shown in Figure 36. This option uses DXF files for creating all layers in the Plane Strain mesh.

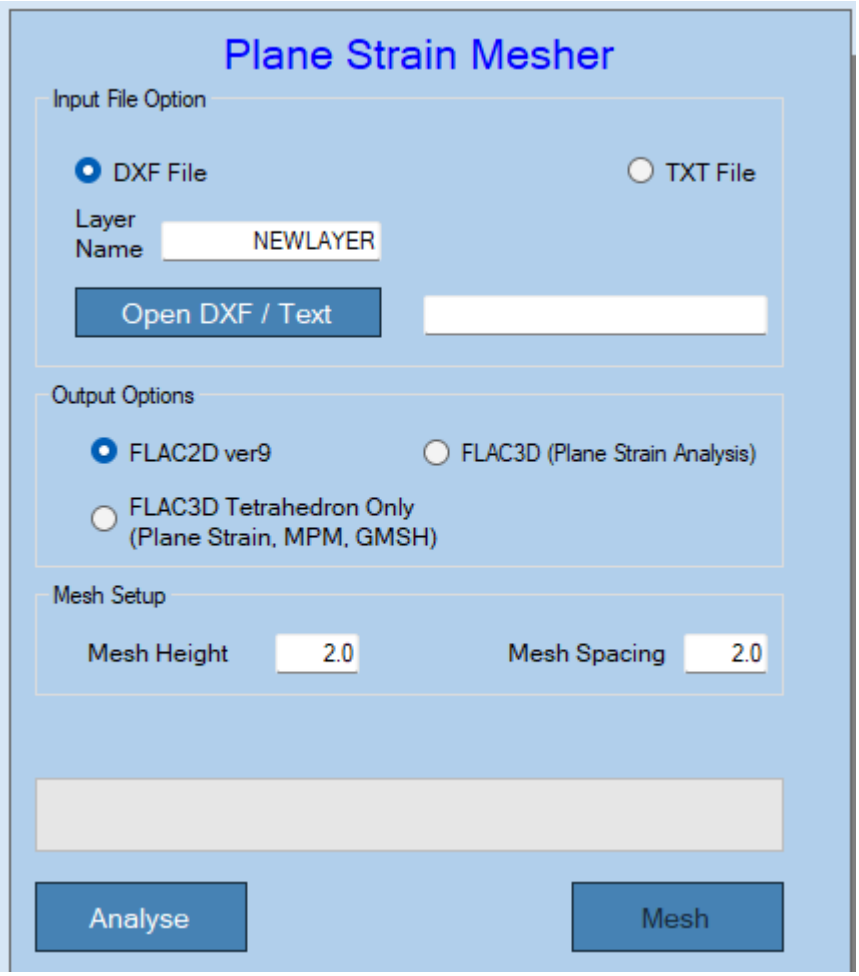


Figure 36: Plane Strain Dialog Window.

- **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. Only geometries defines by POYLINES are accepted, and they need to have at least 2 segments, even for straight lines.

- **Output Options:** Selects the Plane Strain mesh type that will be issued for FLAC3D, FLAC2D or general purposes. In case of FLAC3D, the mesh thickness is 1 element. The option *FLAC3D Tetrahedron Only* issues two meshes; the FLAC3D mesh in the original working folder and a pure tetrahedron mesh located in a subfolder, called TETs. This last one can be opened with MGSH software (free) and it can be used for general purposes such as Material Point Method (MPM) analysis.
- **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 v6* for further editing/refining the Plane Strain mesh (see section 5.1).
- **Analyze**: Runs the analysis of either the CAD or Text file, and display the image shows in Figure 37.

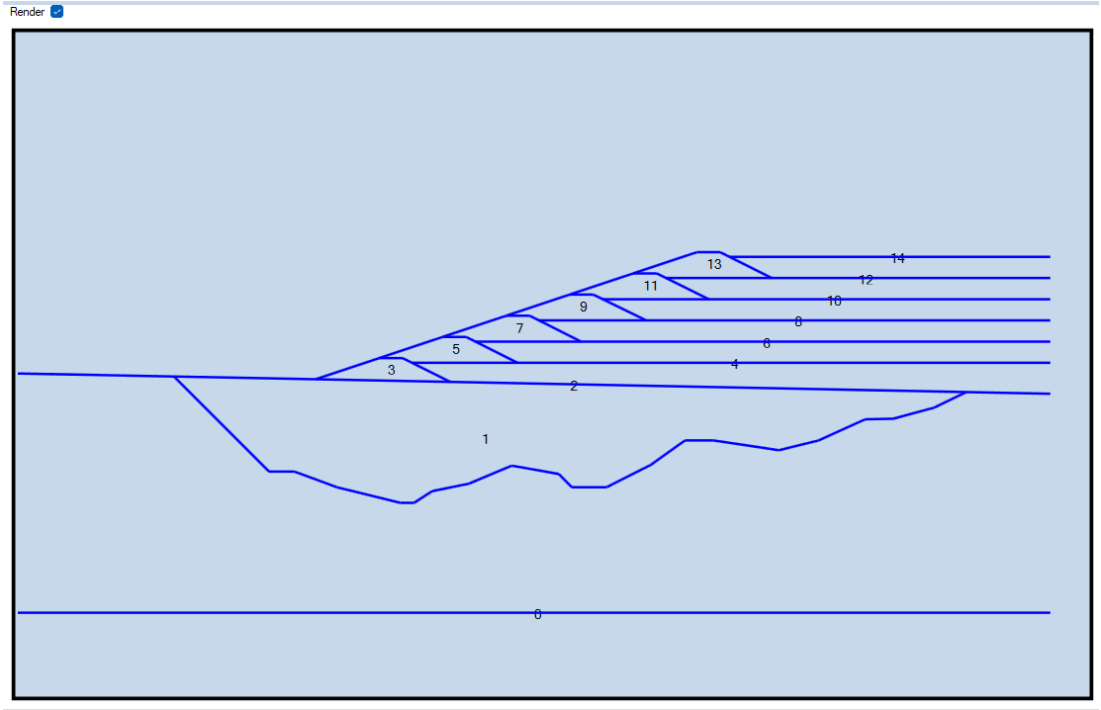


Figure 37: 2D Plane Strain Geometry in Render Area.

The render area of *Geo Meshing v6* 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 36.

6.7. Help Menu

6.7.1. Manual Option

- **Help Menu, Manual**: This option (shown in Figure 38) opens this manual.

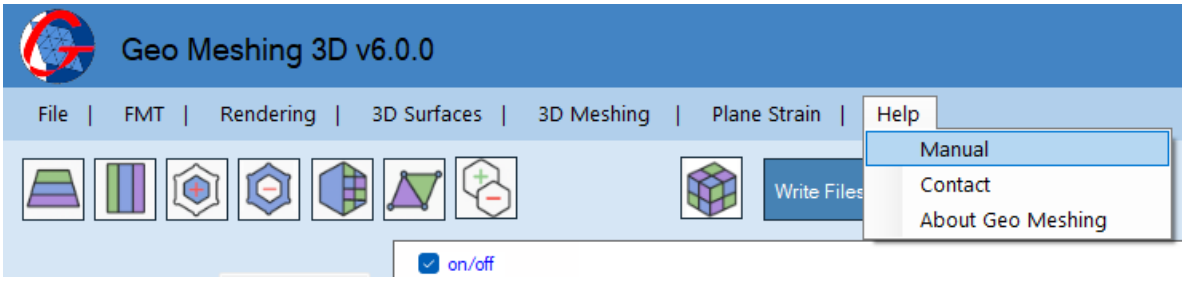


Figure 38: Help Strip Menu – Manual.

6.7.2. Contact Option

- **Help Menu, Contact**: This option (shown in Figure 39) displays support information.

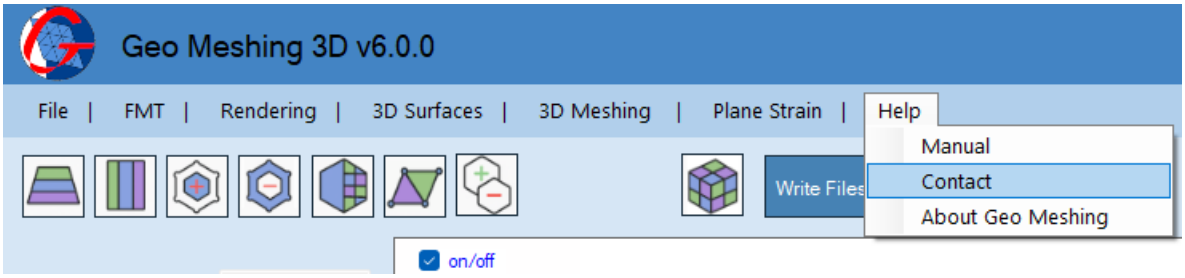


Figure 39: Help Strip Menu – Contact.

6.7.3. About Geo Meshing Option

- **Help Menu, About Geo Meshing**: This option (shown in Figure 40) displays *Geo Meshing v6* information along with the *Geo Meshing v6* code (See section 4.3 for more information).

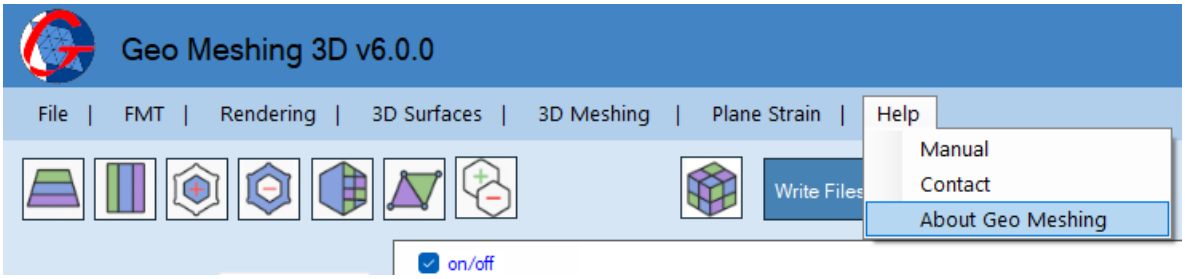


Figure 40: Help Strip Menu – About Geo Meshing.

7. RENDER

The *Render* area is advanced graphical user interface (GUI), that serves the purpose of visualizing any generated landform and 2D geometries. It also serves as GUI for input coordinates.

When displaying 3D landforms, use:

- **Right Mouse Button:** Hold the right mouse button while moving the mouse around to rotate the graphics.
- **Mouse Wheel:** Scroll the mouse wheel to zoom in or out the graphics.
- **Mouse Wheel:** Press the mouse wheel to pan the graphics.
- **Left Mouse Button:** Click the left mouse button to record coordinates over the landform graphics.

7.1. 3D Render Controls

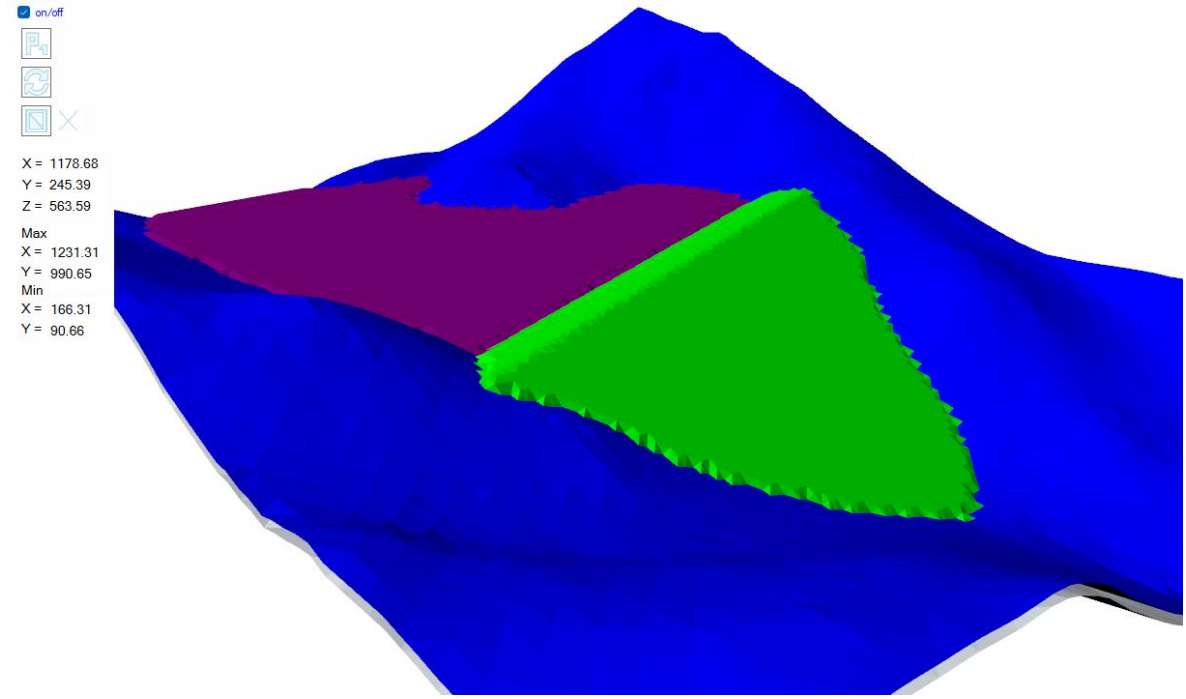


Figure 41: 3D Render Section – Surface Mode.

-  Changes the model projection to perspective view.  Changes the model projection to top flat view.

-  Refreshes the model. Useful for deleting coordinate marks.

-  Alternates between activating/deactivating the mesh over the surface.
  -  Indicates whether the mesh mode is active.

7.2. 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.

7.3. Selecting Coordinates with Mouse

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

- Select one of the options of section 6.4.
- Use the *Rendering* option described in section 6.3, for loading the desire *Geo Meshing v6* (*.gmm or *.gmms) 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 pink. 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.

8. MESH FILES

Producing meshes files for FLAC3D®, FLAC2D®, MPM or NUMGEO is direct and effortless. Once you have generated all the surfaces and you have produced your meshes, just click on the "Write Files", button located at the tool bar, to write the mesh to a file. Once you click on that button the dialog window of Figure 42 will display.

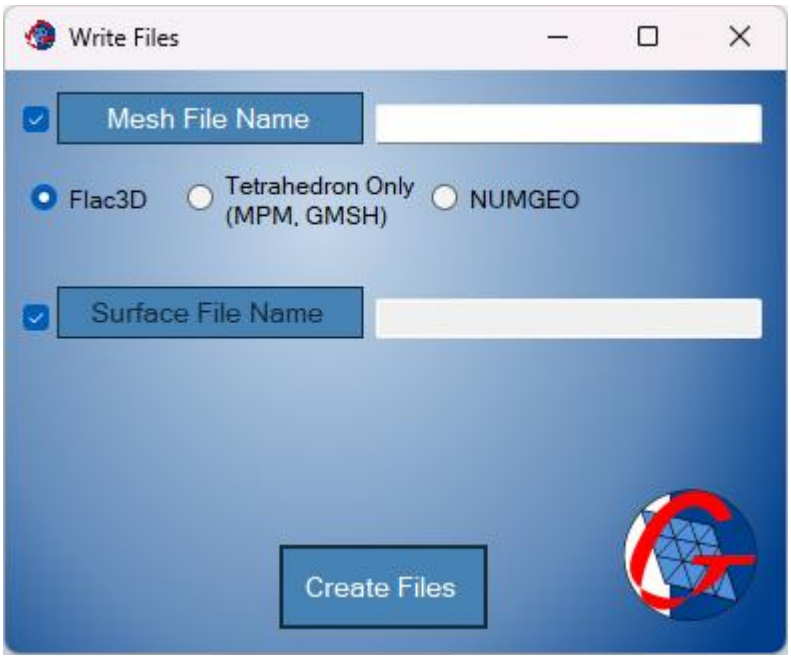


Figure 42: Write Files Dialog Window.

Two independent options can be selected.

- **Mesh File Name:** This button defines the name of the mesh file where the generated 3D mesh is going to be exported. The extension of the file is defined according to the option selected right underneath this button. 3D meshes can be exported for FLAC3D (*_Grid.FLAC3D), Tetrahedron only (*.msh) and NUMGEO (*.inp). These last two options are tetrahedra only based meshes.

- **Surface File Name:** This option saves all the 3D Surfaces the project or the Master Surface. The surfaces will be saved in one files, but each surface is individually saved within the file, so when later restored, it is still separated from the rest.

When the *2D Meshing* option is used, a file named *call_all.dat* is created in the working folder. This *.dat file will call all files needed for producing the 2D mesh.

In order to import a mesh into FLAC3D® or FLAC2D®, 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 v6*. In addition, the output generated by *Geo Meshing v6* is optimized for reducing the number of grid points and zones, thus Flac3D® will not need to merge existing grid points when importing meshes. The same is true for faces on existing zones.

After every time a *3D Surface* option is successfully generated, a file will be created in the working folder, *source_suffix.gmm*. *Source* represents the name of the *Active Surface* over which the 3D meshing option was performed. A suffix is added to the *source* file that helps identifying the type of 3D option operated over the source file.

3D Meshing Option	Suffix
Linear Fill	_LFI
Vertical Fill	_MSE
Polygonal Fill	_FIL
Polygonal Excavation	_EXC
Finer Mesh	_FIN
Interface	_INT
Add	_ADD
Subtract	_SUB

For example, say the *Active Surface* name is *mesh1* and if we perform a *3D Polygonal Fill* over this file, a file *mesh1_FIL.gmm* will be created. Similarly, if we perform a Subtraction over the last *Active Surface*, a file *mesh1_FIL_SUB.gmm* will be created, and so on.

9. TROUBLE SHOOTING

- License file is not working.
  - Get a new license file using the right *Geo Meshing v6* code.
  - You may need administrator privileges to run the program. If you have administrator privileges, please configure *Geo Meshing v6* properties to run as administrator.
  - Provide reading and writing privileges to *C:\Program Files (x86)\Geo Meshing*. *Geo Meshing v6* needs to read and write the license file.
  - 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 v6* by visiting *Geo Meshing* webpage (www.geomeshing.com).

- 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.

10. EXAMPLES

10.1. Example 1 – Plane Strain Mesh for FLAC3D/FLAC2D

10.1.1. Generating the Mesh

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 6.6. Figure 44 shows the layer definition for this example.

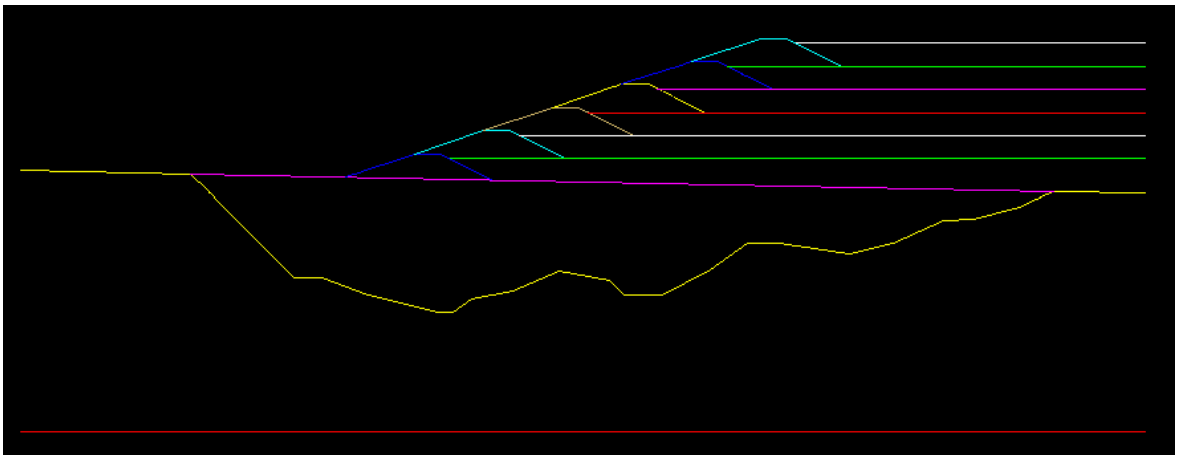


Figure 43: Dam CAD drawing.

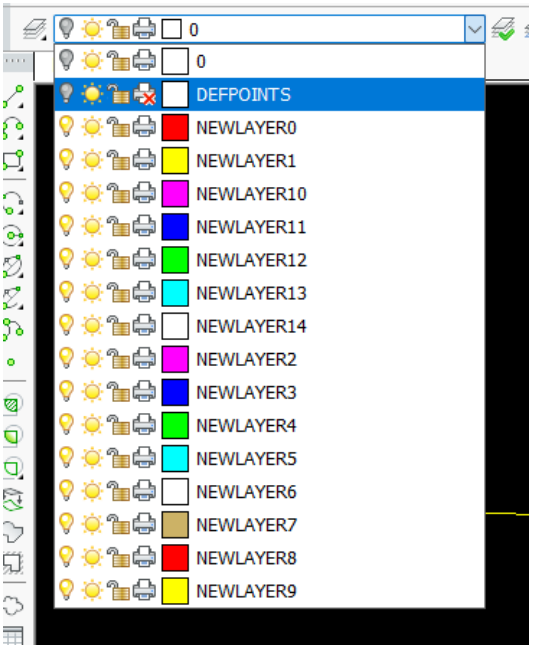


Figure 44: CAD drawing layers.

In the following steps, a red box will indicate the selected option used in *Geo Meshing v6* in the example.

Select *Plane Strain* as shown in Figure 45.

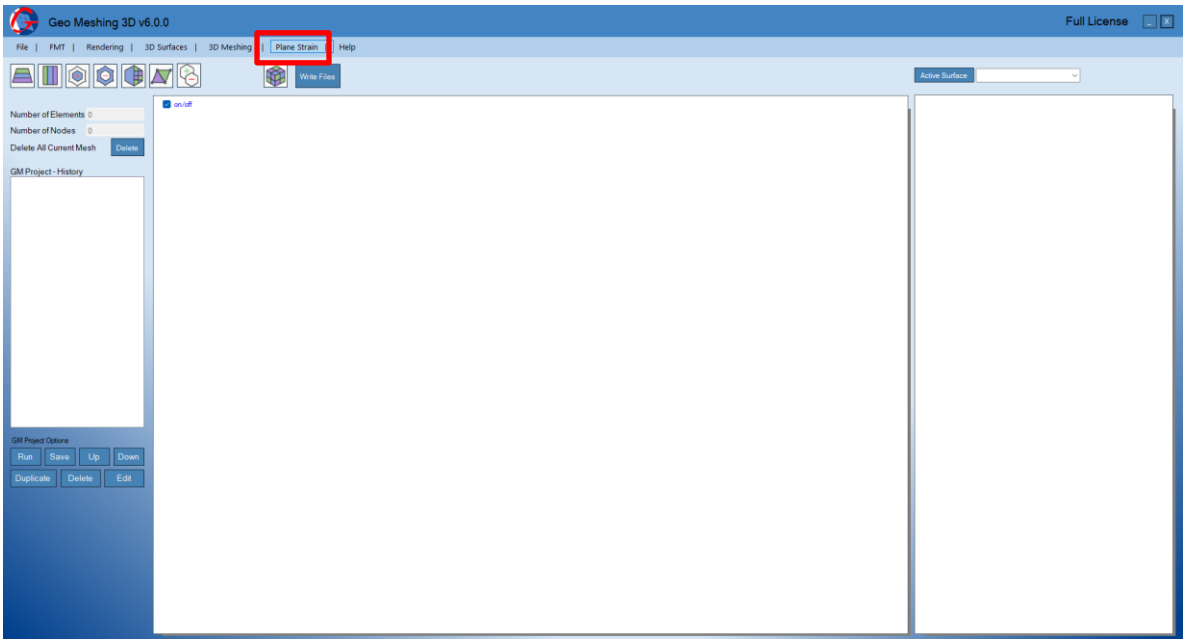


Figure 45: Geo Meshing – Plane Strain.

After selecting the previous option, the dialog window shown in Figure 46 will appear. Notice that by default *FLAC2D ver9* option is selected, but *FLAC3D* (or MPM) options can alternatively selected.

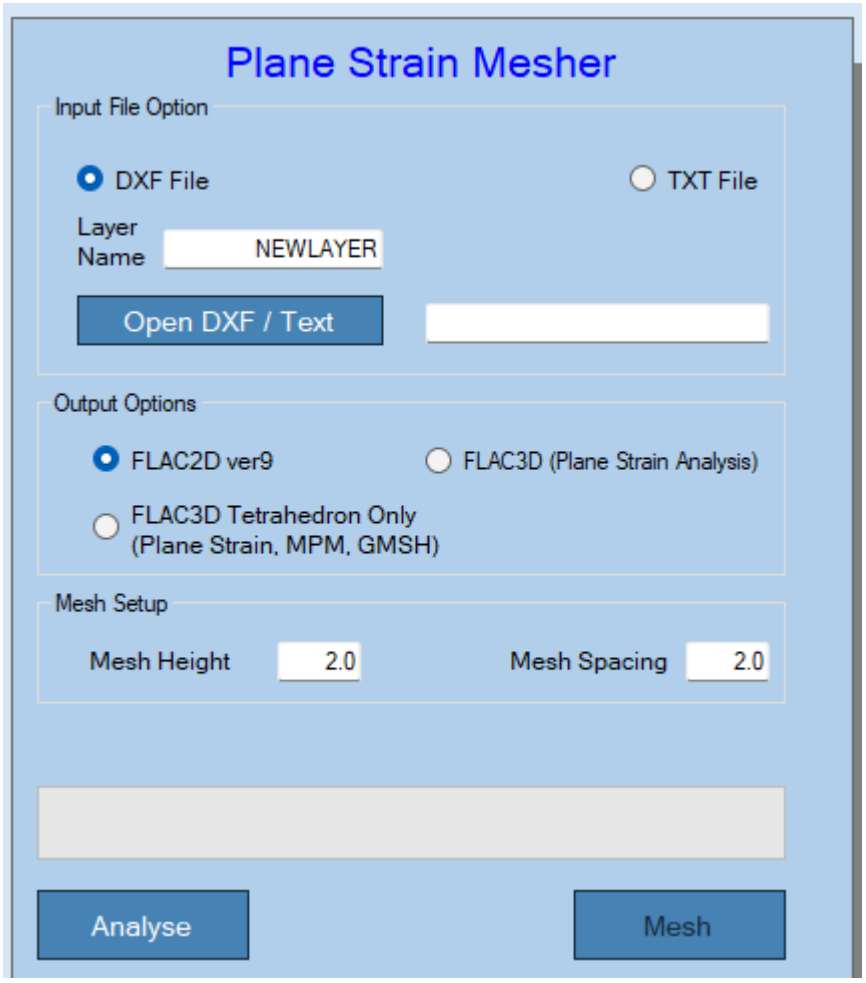


Figure 46: Plane Strain option – Open.

Select DXF file (default option), keep the layer name and click *Open DXF / Text*. The dialog window of Figure 47 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 47.

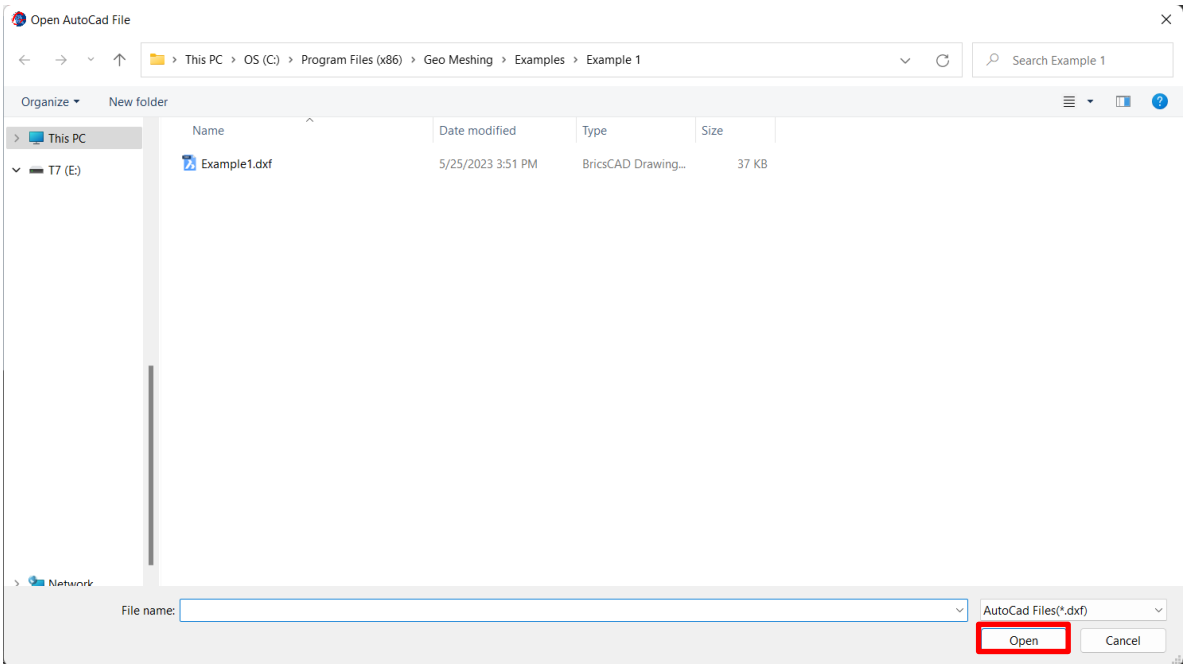


Figure 47: Open CAD file dialog window.

As shown in Figure 48, the name of the DXF file will be displayed next to the *Open DXF / Text* button. Enter the mesh heigh and spacing as shown in Figure 48 and click *Analyze.* The DXF is loaded into *Geo Meshing v6* as shown in Figure 49. Notice that Figure 49 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).

TIP: The polylines defining each part of the geometry must have at least 2 segments.

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 v6.*

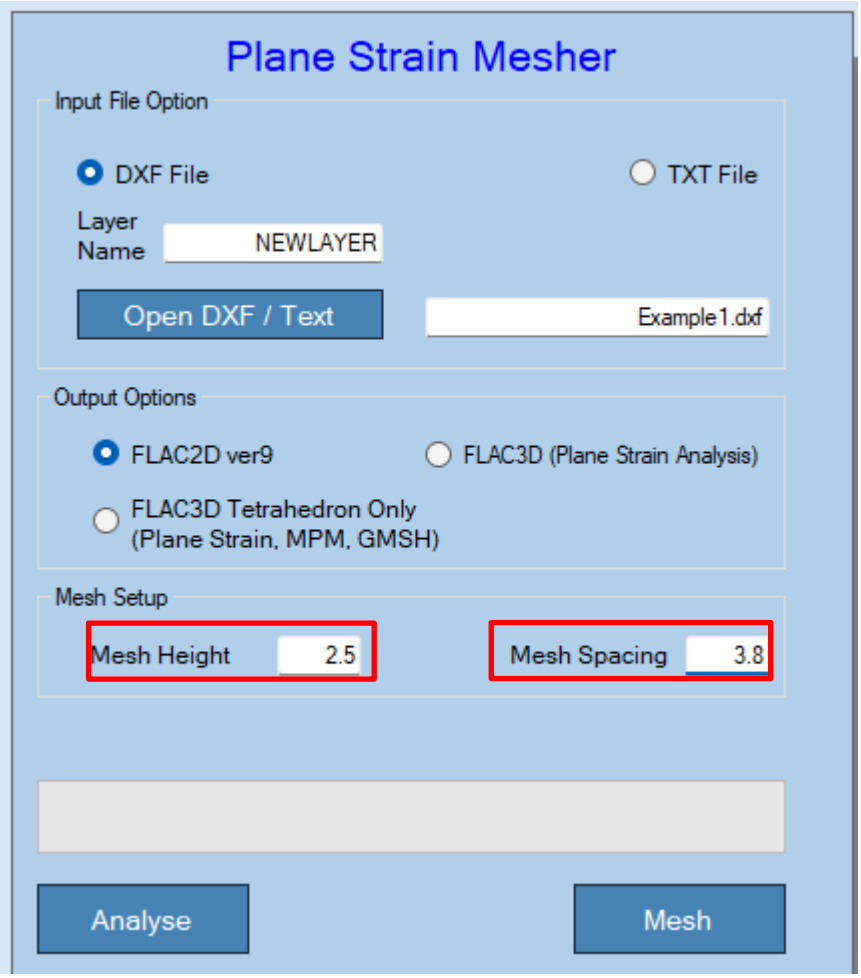


Figure 48: Plane Strain option – Analyze.

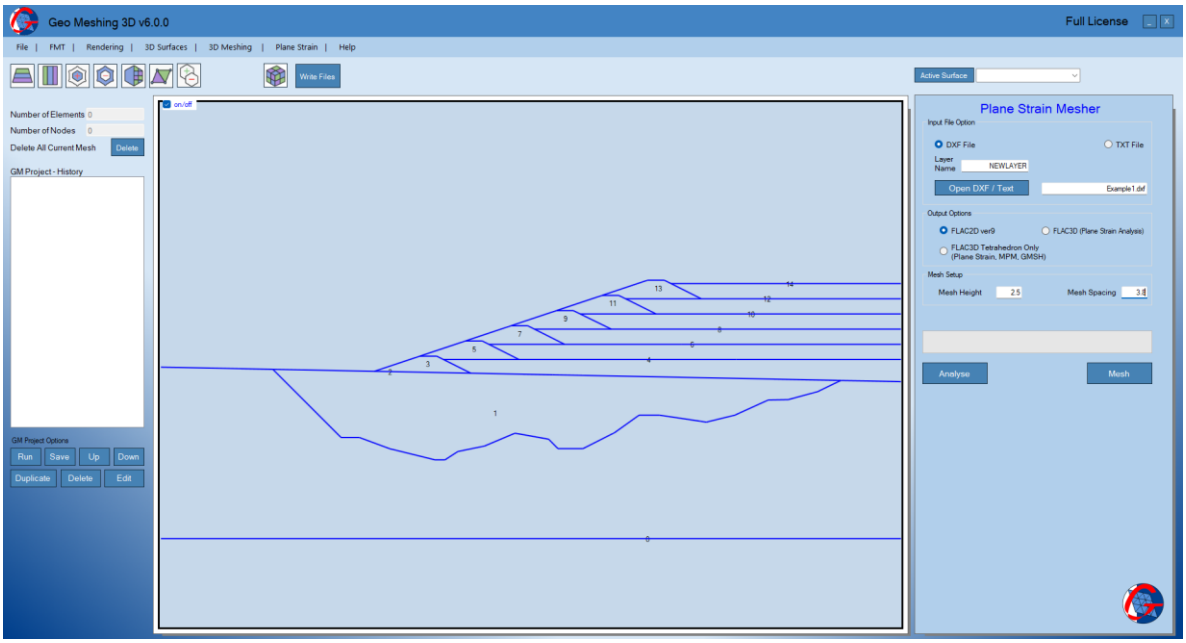


Figure 49: Geo Meshing – 2D wireframe render.

A file called *call_all.dat* will be created In the *Example 1* folder. This file can be loaded into FLAC2D using *control+o* keyboard combination.

FLAC3D or FLAC3D only tetrahedron (MPM) options can also be selected for creating a 3D plane strain model. The Plane Strain models are shown for FLAC2D and FLAC3D in Figure 50 and Figure 51, respectively.

Finally, Figure 52 shows one of the many features of *Geo Meshing v6*, automatic layering. During the meshing process, *Geo Meshing v6* produces horizontal layers for each of the "New Layers" defined in the DXF file, name them sequentially and store them in *slot 2*. Notice that not all the layers name are shown in the legend.

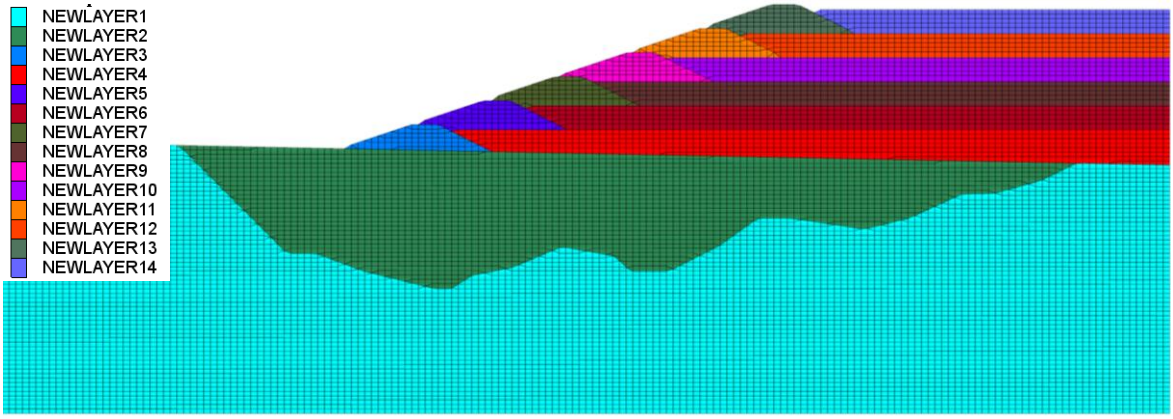


Figure 50: FLAC2D Plane Strain model.

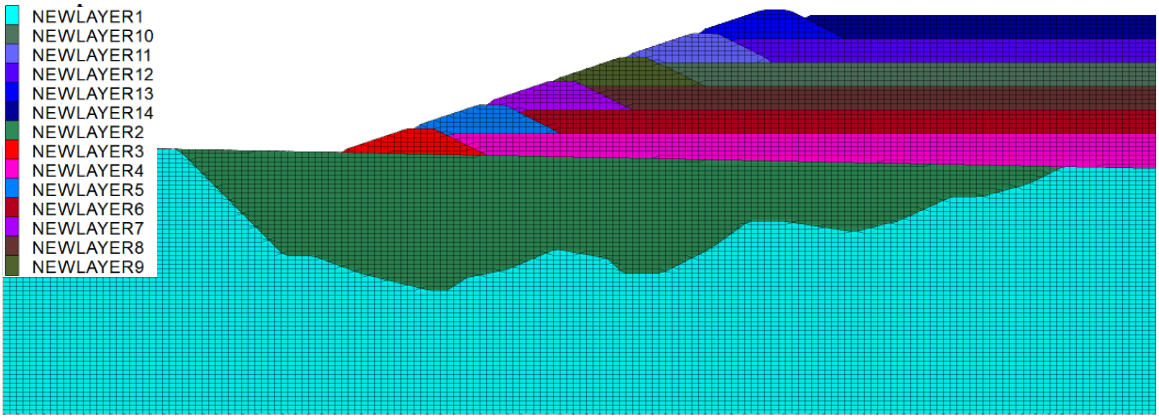


Figure 51: FLAC3D Plane Strain model.

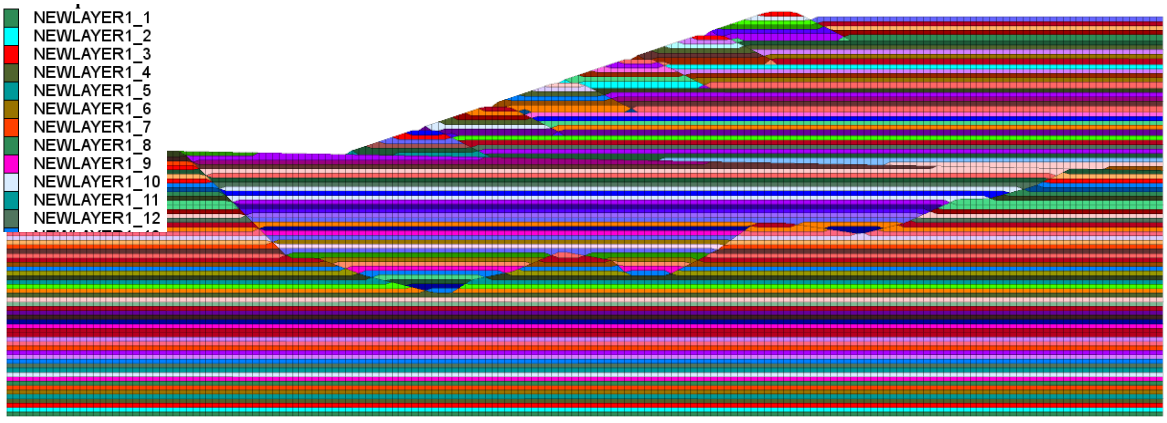


Figure 52: FLAC2D / FLAC3D Automatic Layering.

10.1.2. Editing the Mesh – The History file

Most of the events generated in *Geo Meshing v6* are recorded in the history file, when the file has been properly setup (section 5.1). This file can be edited in *Geo Meshing v6,* for further tailoring /editing meshes. In the following example, the previously generated Plane Strain mesh (section 10.1) is going to be edited. This same procedure can be applied to full 3D meshes.

After the mesh process has been completed (section 10.1.1), *Geo Meshing v6* window looks like the image displayed in Figure 53. Click Save and *Geo Meshing v6 History File* windows will pop up and select *Create*, as shown in Figure 53. Navigate to where Example1.dxf file is located (C:\Program Files (x86)\Geo Meshing\Examples\Example 1) and save the history file with the name Example1.*gmp* as shown in Figure 54.

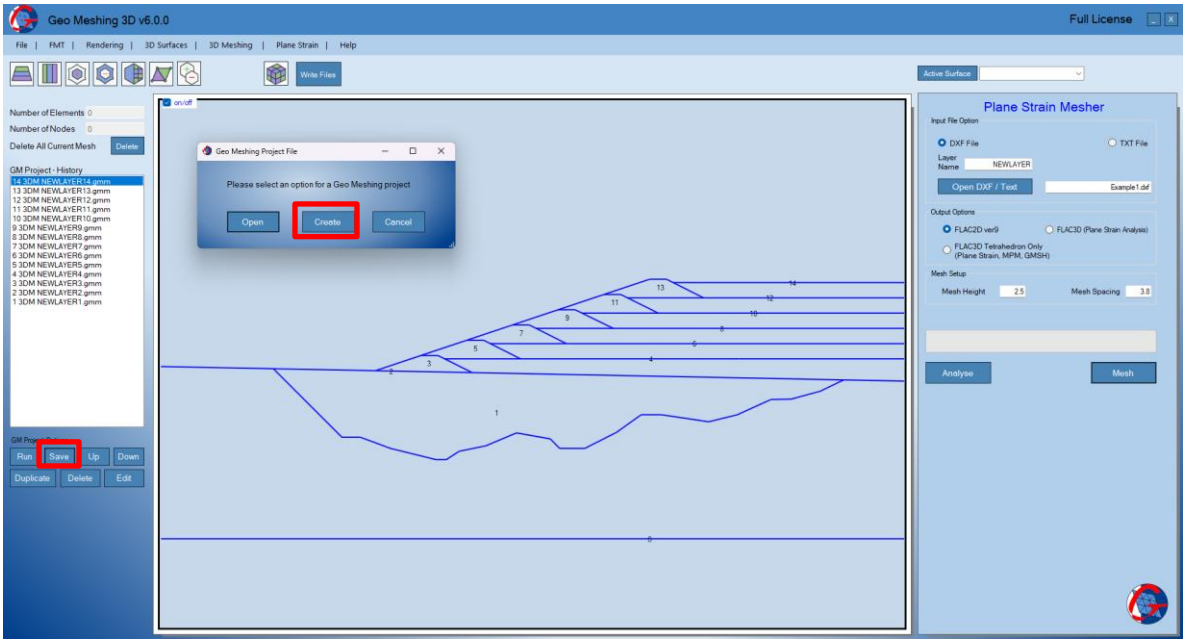


Figure 53: Project History Edit.

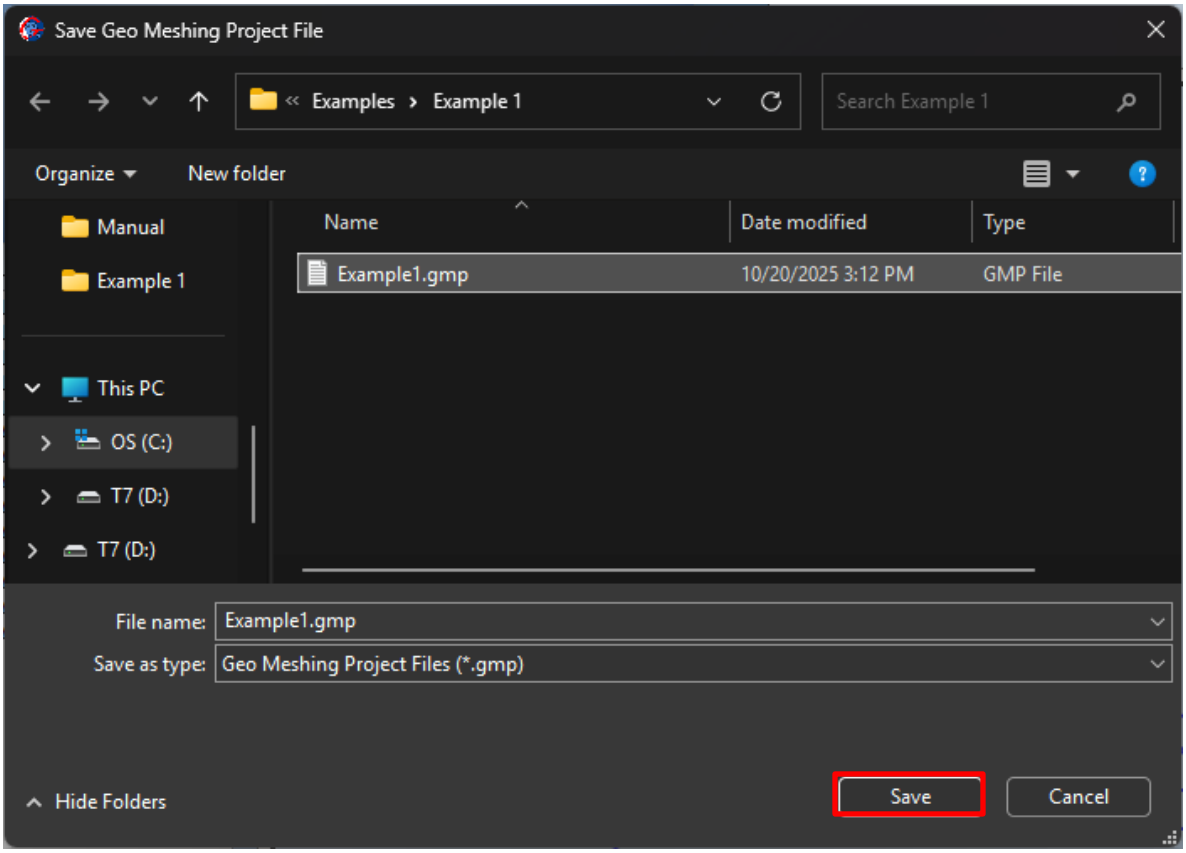


Figure 54: Save Geo Meshing History File.

Click *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 10.1.1.

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.

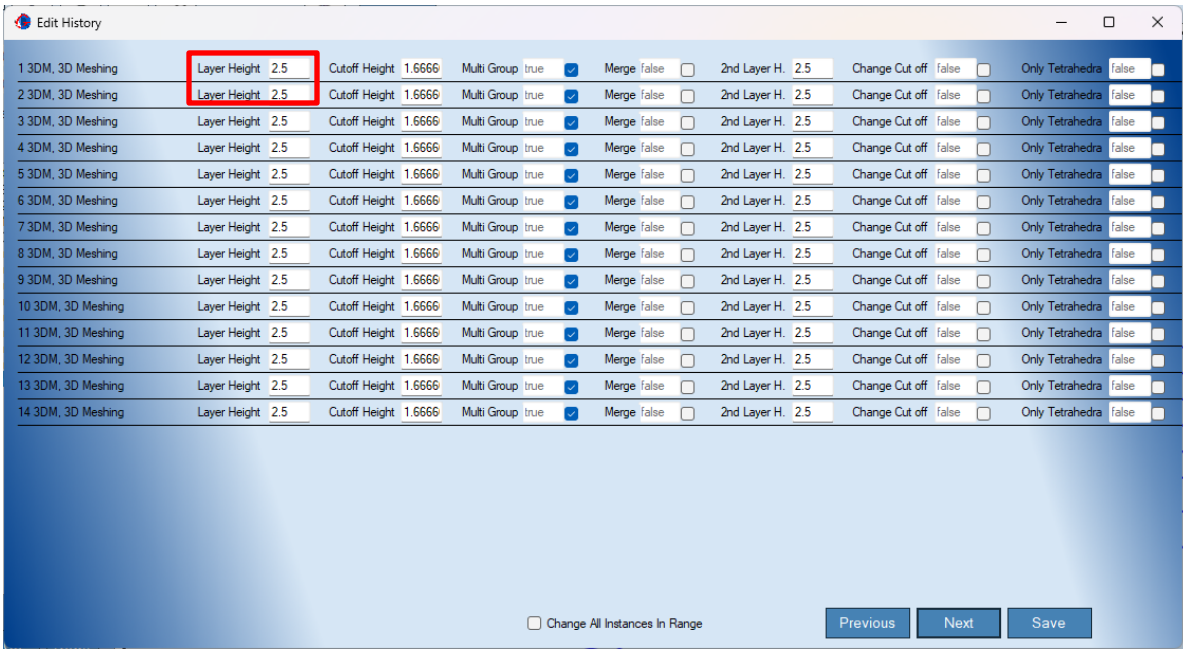


Figure 55: Geo Meshing Edit Windows.

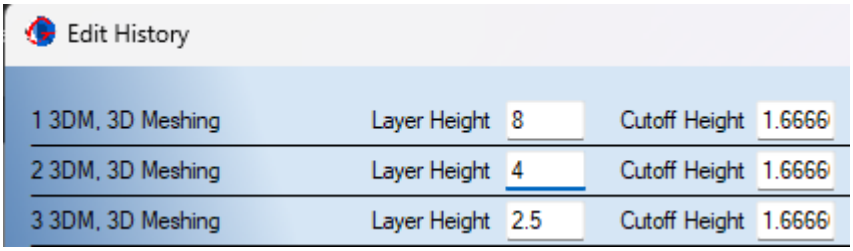


Figure 56: Layer Height Edited.

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*.

Notice that the process runs outside of the *Plane Strain* environment, thus the mesh is treated as a full 3D mesh, even though it is only a 3D slide.

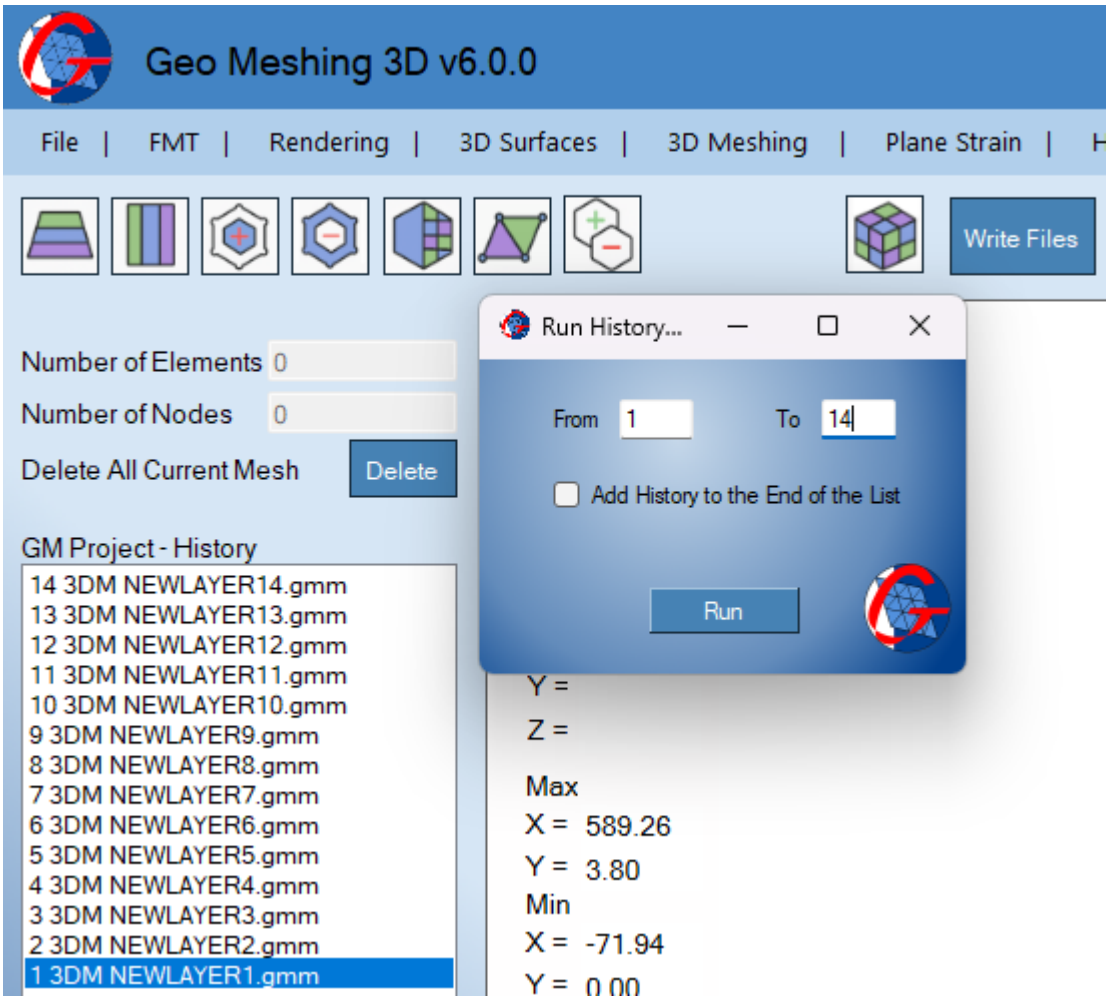


Figure 57: Run History.

Once the process has been completed, *Geo Meshing v6 GM History* box will display the same 14 events but updated with the new options. If you wish you can save the project file at this point. To save the file just click the *Save* button underneath the project history file. Figure 58 shows how the 3D surfaces look after the 14 events run.

Note that the model is a FLAC3D mesh, so we need to convert it back to FLAC2D format. First write the mesh to a FLAC3D file using "Write Files" button. Define a name for the _Grid.FLAC3D file, click Save and then click Creates Files. Second, go to File>Tool>FLAC3D to FLAC2D and convert the just saved FLAC3D file to FLAC2D file for the y = 0.0 plane.

Figure 59 shows the full FLAC2D Plane Strain model. You can load this model following the instructions described in the previous section. Notice that the mesh is much coarser at the base but still has a smaller size at the foundation surface. The different between the edited model and the original can be observed by comparing Figure 59 with Figure 50.

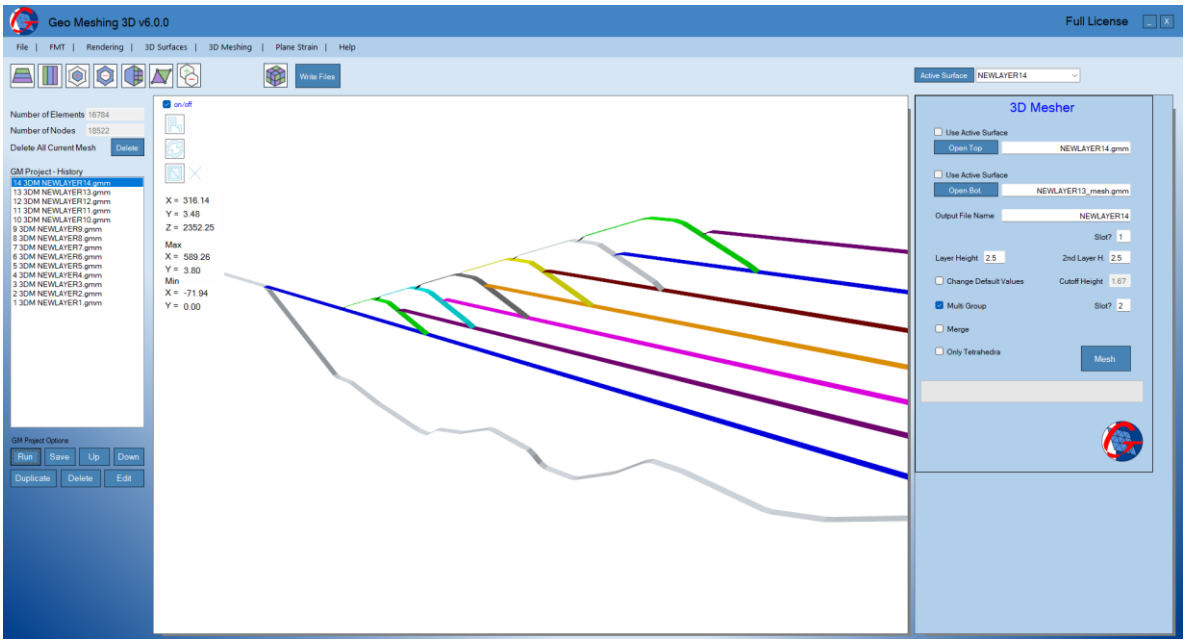


Figure 58: Geo Meshing – After the 14 events run.

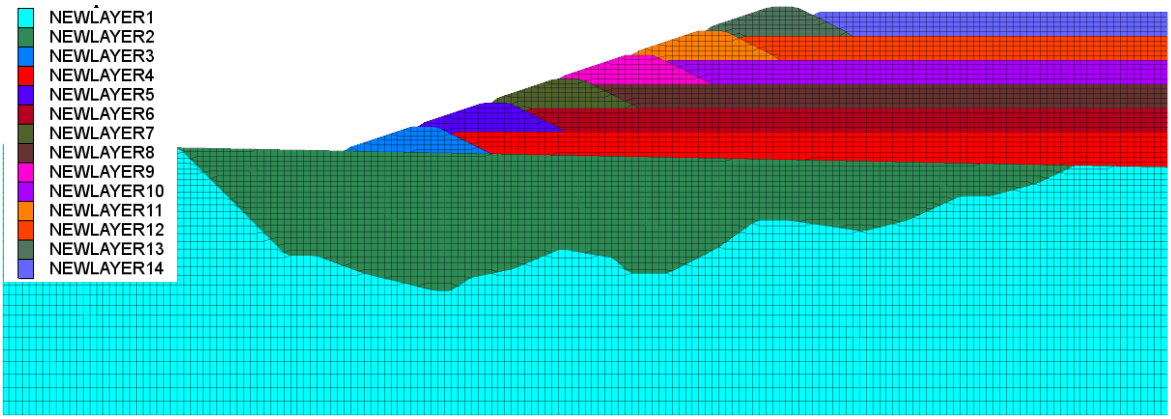


Figure 59: FLAC2D Plane Strain model. Coarser Base.

10.2. Example 2 – Using Geo Meshing for FLAC3D (3D Model)

This example has three parts; First it shows how to generate a topography base with *Geo Meshing v6*, and then, it explains how to create two types of 3D surfaces with *Geo Meshing v6.* The tools used in this example for the following 3 sections are *FMT*, *3D Polygonal Fill* surface and *3D Linear Fill* surface, respectively.

10.2.1. Create a Topographic Base, First Part

Locate the folder Example 2 (C:\Program Files (x86)\Geo Meshing\Examples\Example 2) and create a new text file. This file defines the plain view domain of the topographic base. Name this file as *domain.txt* and write the two lines shown in Figure 60. Save the file.

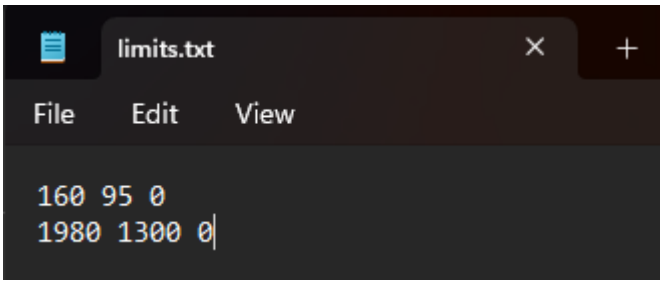


Figure 60: Text file for Model limits.

Open *Geo Meshing v6* and click on the FMT option. A windows will emerge inquiring about the project history file. Select *Create*, make sure you are creating the new project history file in the *Example 2* folder. Click *Save*. A project history file has been created at the working folder.

In the *Flat Mesh & Topography* (FMT) window select *Text File* option and click *Open Points Cloud* and select *domain.txt* file. Finally, input 12 units for *Delta X* and click *Mesh/Interpolate*. Go to the working folder, it should has now the file *domain_mesh.gmm*. Left hand of Figure 61 shows how FMT window should look like, after selecting the previous options.

Go back to FMT windows and now select *CAD file*, click *Open Points Cloud*, and a new window *CAD Import DXF* will emerge. Click *Open CAD File* and select Example2.dxf and click *Open*. From the dropdown menu select PRUEBA layer and

click *Generate Point Data*. Figure 62 shows the *CAD Import DXF* windows with the selected options. Back again in FMT window and unselect *Flat Mesh Gen* and click on *Open Mesh*. Select *domain_mesh.gmm* and click *Open*. Select *Interpolate Z*. Finally, input 24 in the *Minimum Thickness* text box and click *Mesh/Interpolate*. The right-hand image of Figure 61 shows the FMT windows with the selected options. Warning! This operation might take some time.

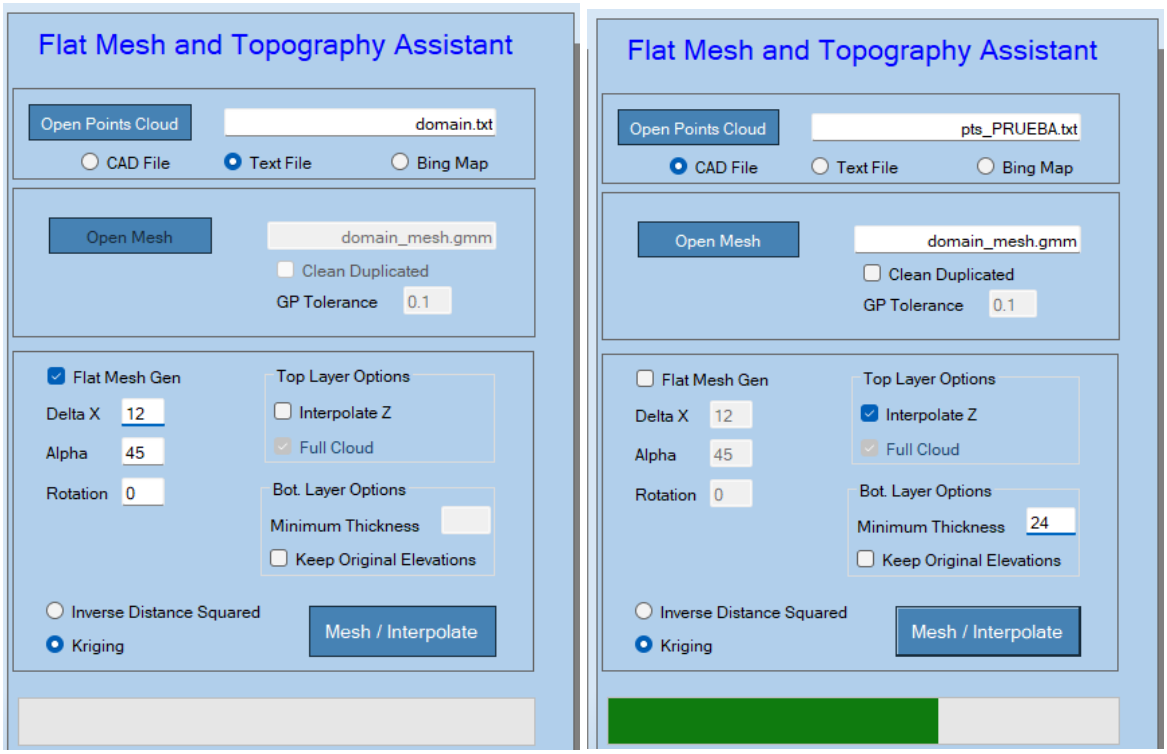


Figure 61: FMT options.

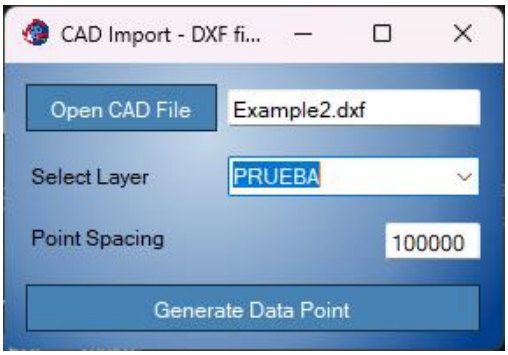


Figure 62: CAD Import DXF Window.

In the *3D Meshing* window input "Base" in the *Output File Name* text box. Then input 24 units in the *Layer Height* text box and input 12 units in *2nd Layer H.* text box. Click *Mesh*. Figure 63 shows the 3D Meshing window with the selected options.

The final surface in *Geo Meshing v6* and FLAC3D are shown in Figure 64. Notice that in order to export the mesh for FLAC3D, the "Write Files" option must be used first.

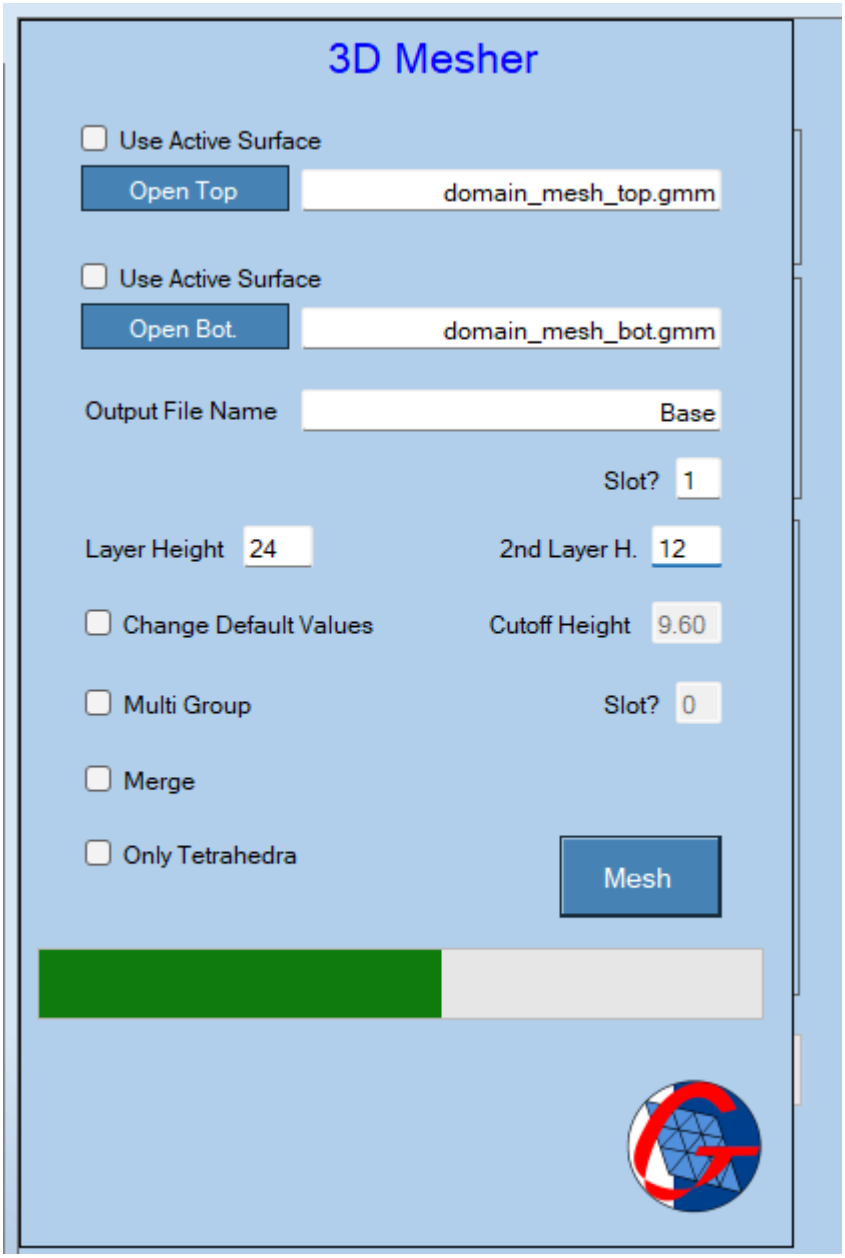


Figure 63: 3D Meshing Window – Base 3D Model.

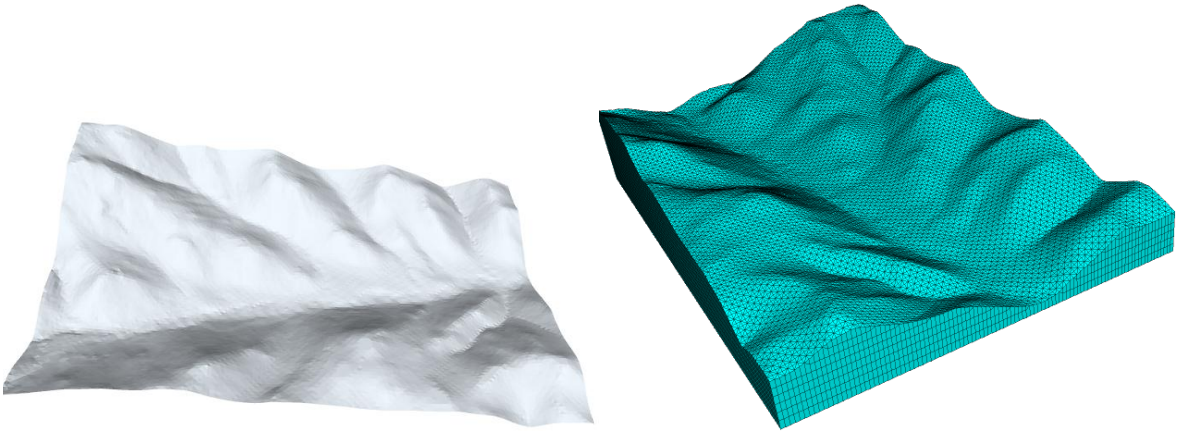


Figure 64: 3D Topographic Models.

10.2.2. Polygonal Fill Example, Second Part

This part of the example illustrates how to create a polygonal fill, using the Active Surface Base.

Go to the *Geo Meshing v6* tool bar and click on  (*3D Polygonal Fill Surface*). If the *Geo Meshing v6* project history file windows is prompted, select the *Example2.gmp* file created in the first part of Example 2.

Make Sure *Base* is selected in the Active Mesh dropbox. Click the P1 icon at the top left corner. The surface projection changes as shown in Figure 65. Click over the topographic surface, in clockwise direction, in order to generate the six points shown in Figure 65. As you click on the surface, the coordinates are recorded in the *3D Polygonal Fill* window. When generating the points, one can start from any corner, and it only needs to be located at similar positions. The first point is demarked in red, and the remaining are demarked in pink. If you make a mistake, click *Reset* in the *3D Polygonal Fill* window, and start over. Once the points are defined, go back to the *3D Polygonal Fill* window and input 600 units for the elevation, in the *Fill Elevation* text box, input 300 in the *Fill Horizontal Extension* text box and 1 in the *Fill Slope* text box. Click *Generate*.

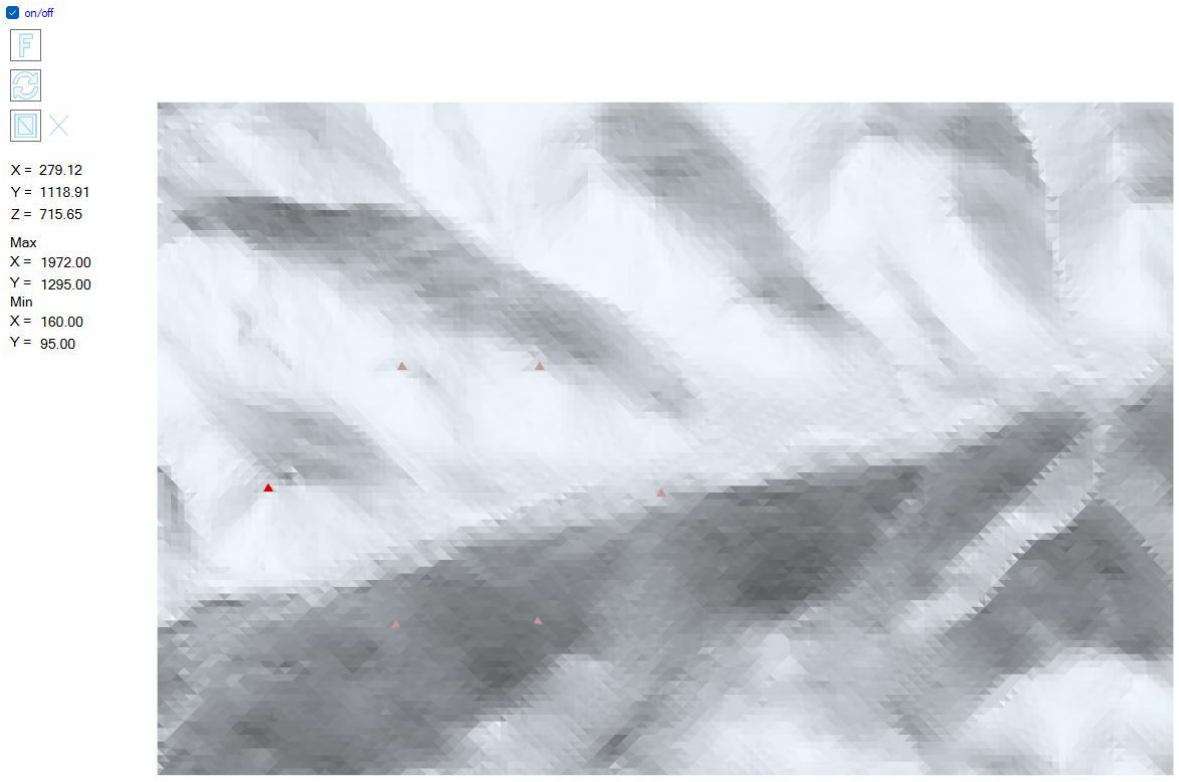


Figure 65: 3D Topographic Surface in Geo Meshing v6.

In the emerged *3D Mesher* windows input *Fill1* for the *output File Name*, input 12 unit for the *Layer Height*, select *Multigroup*, and click *Mesh* as shown in Figure 66.

Click again  icon and click the P1 icon at the top left corner. Once again, click over the topographic surface, in clockwise direction, in order to generate the six points shown in Figure 67. If you make a mistake, click *Reset* in the *3D Polygonal Fill* window, and start over. Once the points are defined, go back to the *3D Polygonal Fill* window and input 700 units for the *Fill Elevation*, 300 for the *Fill Horizontal Extension* and *Fill Slope* of 0.9. Click *Generate*.

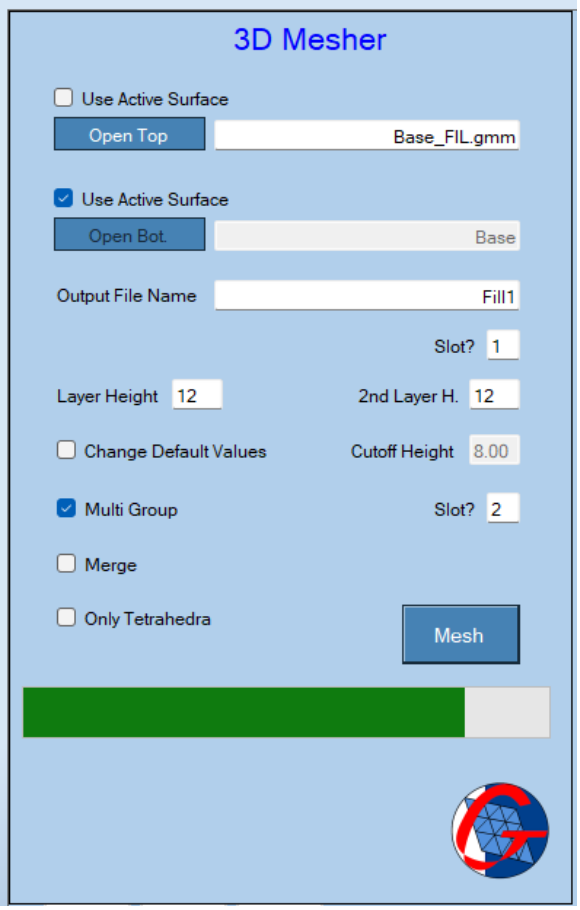


Figure 66: 3D Meshing Window – Fill1 3D Model.

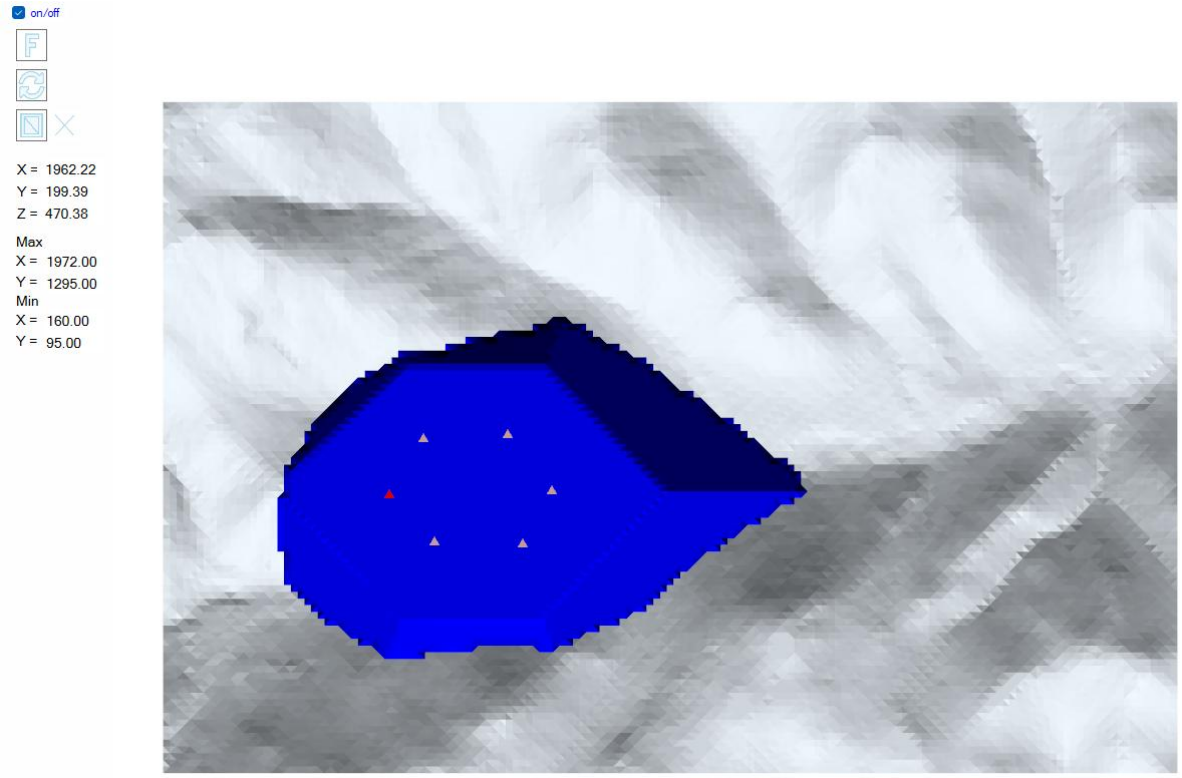


Figure 67: 3D Topographic Surface and 3D Fill Surface in Geo Meshing v6.

In the emerged *3D Meshing* input *Fill2* for the *output File Name*, input 12 units in the *Layer Height* text box. Tick the box next to *Multi Group* and click *Mesh*. Figure 68 shows the *3D Meshing* window with the selected options.

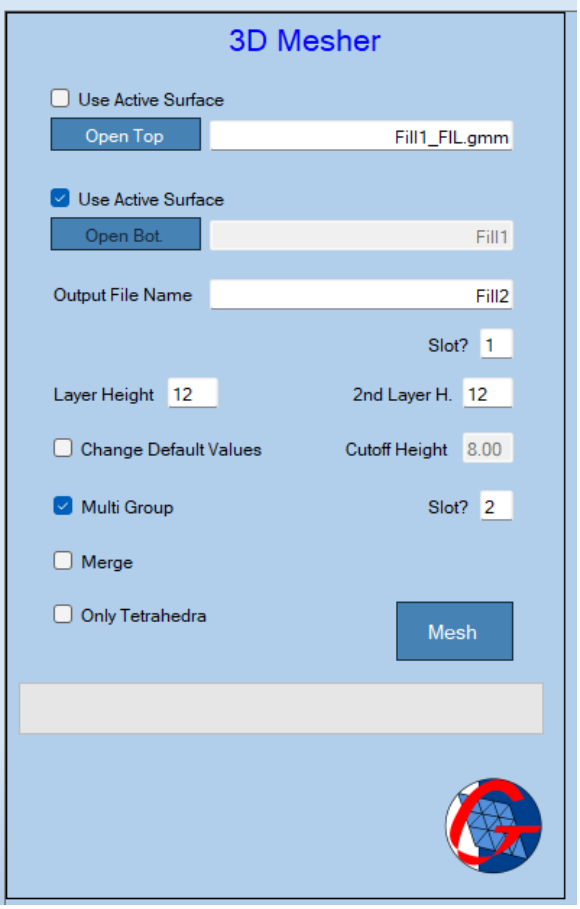


Figure 68: 3D Meshing Window – Fill2 3D Model.

Finally, save the files in a Flac3D file. Name the file as Fills_Grid.FLAC3D and run "Fills.dat" file from FLAC3D. The final surface in *Geo Meshing v6* and the imported mesh in FLAC3D are shown in Figure 69.

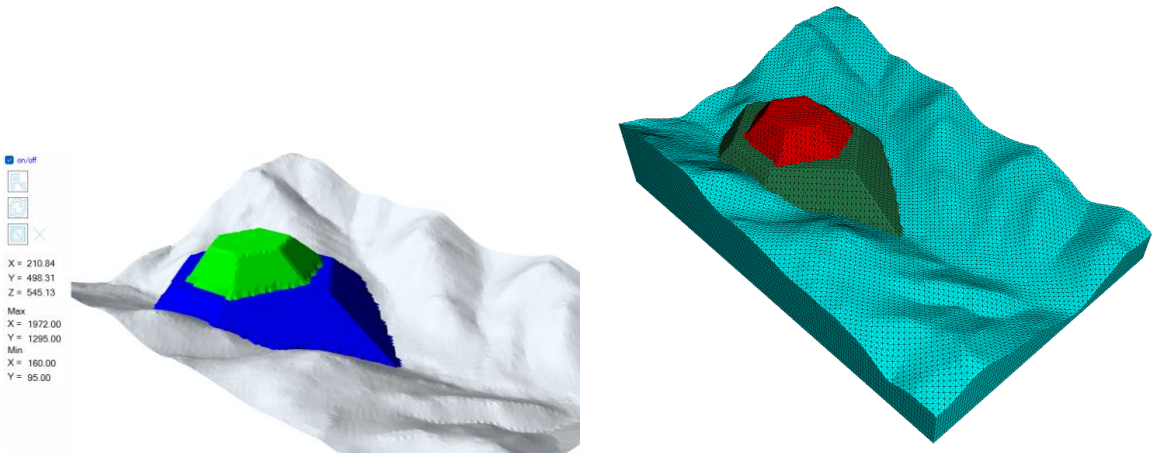


Figure 69: 3D Topographic and 3D Polygonal Fill Models.

10.2.3. Lineal Fill Example, Third Part

The third part of this example shows how to add a ramp to the fill models created in the second part of the example.

Go to the 3D Surface Tool Bar in *Geo Meshing v6* and select  *(3D Linear Fill Surface)*. If the *Geo Meshing v6* project history file windows is prompted, select the *Example2.gmp* file created in the second part of Example 2. If the Surfaces are not loaded, open the Example2.gmp project file.

Make sure that the selected *Active Mesh* is set to Fill2. Click the P1 icon at the top left corner to make a F view. The surface projection changes as shown in Figure 70. Click over the landform surface, in order to generate the points shown in Figure 70. As you click on the surface, the coordinates are recorded in the *3D Linear Fill* window. When generating the points, one can start from any corner, and it only needs to be located at similar positions. The first point is demarked in red, and the remaining are demarked in pink. If you make a mistake, click *Reset* in the *3D Linear Fill* window, and start over.

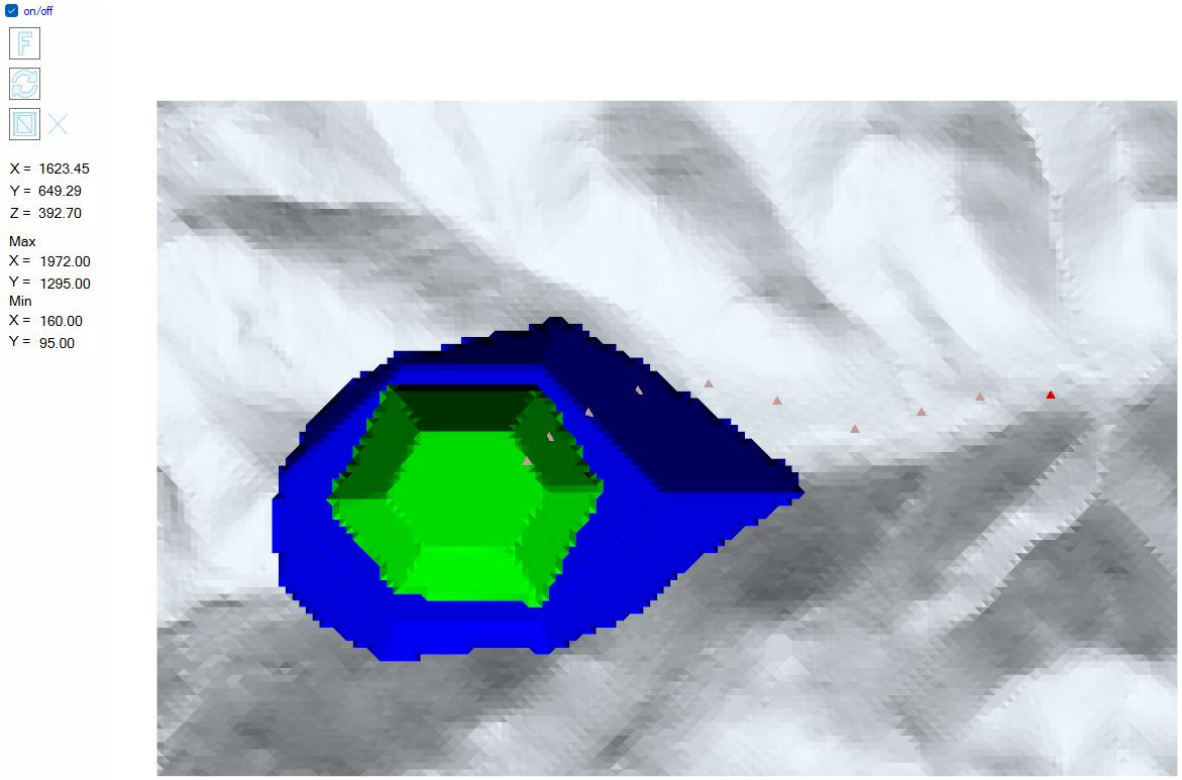


Figure 70: 3D Topographic Surface and two 3D Fill Surfaces in Geo Meshing v6.

Once the points are defined, go back to the *3D Linear Fill* window and unselect the *Horizontal Surface* option. Then check the left tick mark in front of the first coordinate and check both tick marks in front of the last coordinates. Make sure the elevation of the last coordinate is 700 units. Input 1.0 for *Fill Slope 1* and *Fill Slope 2* text boxes and 50 for *Line Offset* text box. Click *Generate*.

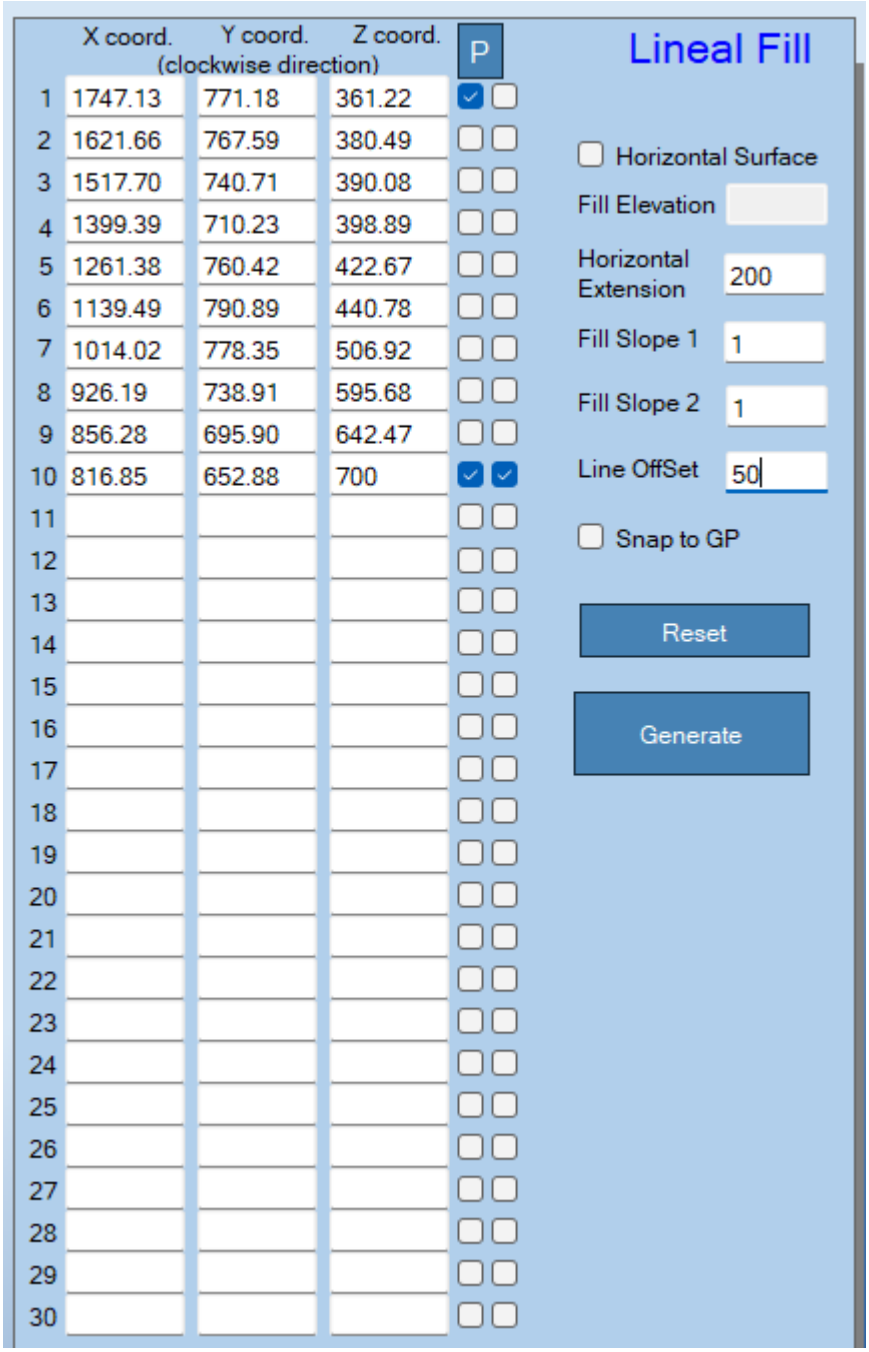


Figure 71: 3D Linear Fill Surface dialog window.

In the emerged *3D Meshing* window input "Ramp" in the *Output File Name* text box. Then input 12 units in the *Layer Height* text box. Tick the box next to *Multi Group* and click *Mesh*.

Finally, export all the mesh to a file and call it from FLAC3D. The final surface in *Geo Meshing v6* and the imported mesh in FLAC3D are shown in Figure 72.

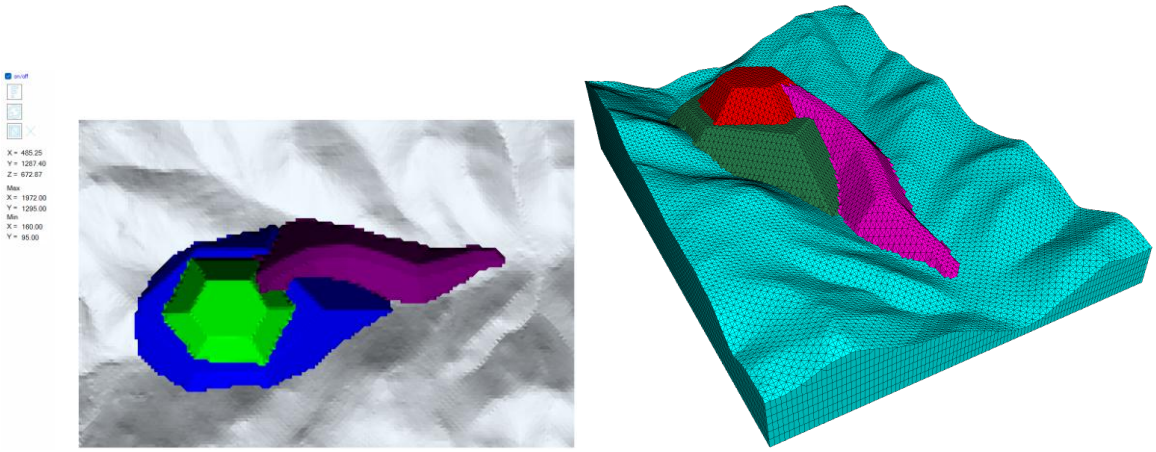


Figure 72: 3D Topographic Model, 3D Polygonal Fill Models and 3D Linear Fill Model.

10.2.4. Closure

Finally, the user is encouraged to try different options in all the tools used in this example (*FMT*, *3D Polygonal Fill, 3D Linear Fill* and *3D Meshing*). For example, try different values of elevation and slopes ratios in the *3D Polygonal Fill* tool or try different *Line Offset* values in the *3D Linear Fill* tool. By trying different option combinations, that are just a click away, the user will discover the real potential and flexibility of *Geo Meshing v6*.

10.3. 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/hIn210Lae-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

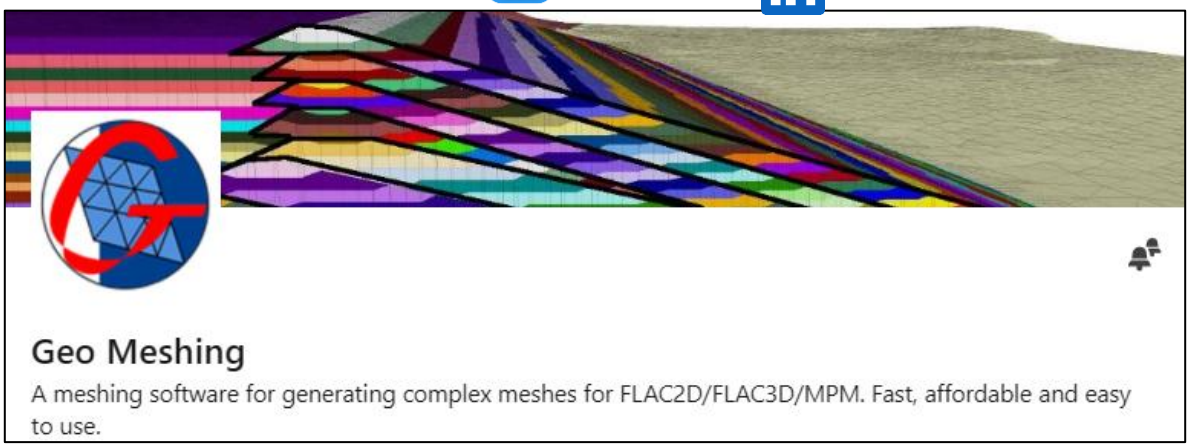
11.CONTACT & SUPPORT

For questions, feedback, or technical support, please contact:

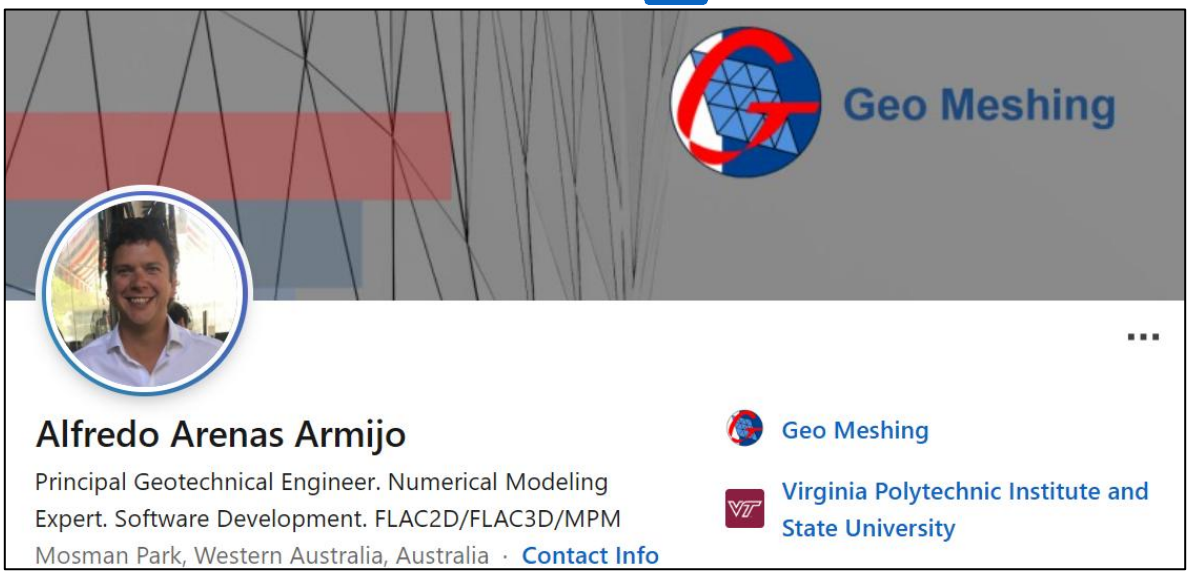
Geo Meshing Team / Email: geomeshing@gmail.com

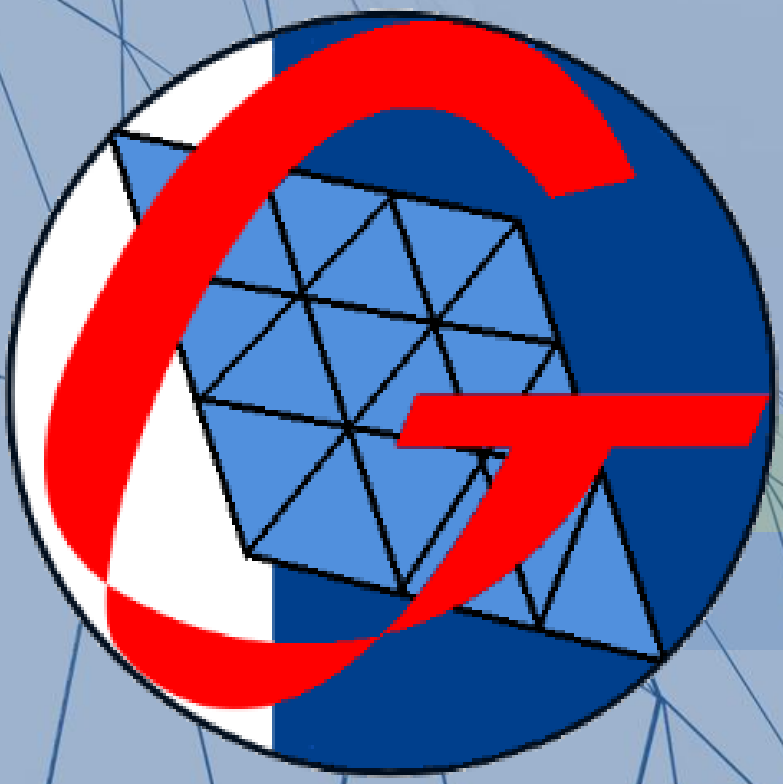
Website: https://www.geomeshing.com

LinkedIn: Geo Meshing



LinkedIn: Alfredo Arenas Armijo





Geo Meshing