# Geo Meshing v3.2

December 2020

# Contents

# 1. Introduction

*Geo Meshing v3.2* is a software specifically designed for developing 3D meshes for FLAC3D® and 2D meshes for FLAC®.

*Geo Meshing v3.2* greatly simplifies the complex process of producing advanced 3D meshes for FLAC3D®, and therefore, reduces the cost associated with those tasks. Likewise, *Geo Meshing v3.2* effortless produce meshes for FLAC®, automatically computing grid elements number, ratios and geometries.

Although *Geo Meshing v3.2* 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 v3.2* 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 v3.2* creates, as much as possible, balanced meshes, where large contrasts in element sizes and orientations are avoided.

- **Parametric body definition**: *Geo Meshing v3.2* uses a parametric definition for creating a wide variety of bodies without the need of a CAD assistant software.

- **CAD compatibility**: *Geo Meshing v3.2* can extract polyline information from DXF files. Using this feature, a topographic landform can be developed in minutes.

- **Bing Maps® Ready:** *Geo Meshing v3.2* can extract topographic surfaces directly from Bing Maps®, thus landforms can be easily imported into *Geo Meshing v3.2.*

- **GID® compatibility**: *Geo Meshing v3.2* reads and imports 2D files created with GID®. These files can be used to produce 3D meshes.

- **Meshing refinement**: *Geo Meshing v3.2* can refine meshes at any level within the model.

- **Interpolation algorithms**: *Geo Meshing v3.2* includes two interpolation algorithms, inverse distance squared and Kriging for obtaining smooth surface profiles.

- **Flexible**: *Geo Meshing v3.2* output files can be post processed and used with other software.

- **Multiprocessor**: *Geo Meshing v3.2* is a parallel processing software, allowing for faster development of meshes.

- **Export to FLAC3D® Geometry:** Any surface developed with *Geo Meshing v3.2* can be transformed and imported into FLAC3D® as a 3D geometry.

## 2.2. 2D Features

- **CAD compatibility:** *Geo Meshing v3.2* can extract polyline information from DXF files. Using this feature, grid size, ratios, elements sizes and model domain can be determined.

- **Flexible:** *Geo Meshing v3.2* allows changing element size across the domain, thus specific areas can have a greater resolution. Ratios for achieving refinement are internally computed.

- **Easy to edit:** *Geo Meshing v3.2* generates a *.dat file that can be called from FLAC®. This file can be easily edited with any text editor for further analysis/ edition.

# 3. *Geo Meshing v3.2* Setup

*Geo Meshing v3.2* can be installed automatically or manually. It is highly recommended to install it automatically.

Once *Geo Meshing v3.2* 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.

> *Warning: When using Geo Meshing v3.2, do not use folders and/or file names with spaces for your data files.*

For example:

*C:\Users\user1\Documents\testing geo meshing*

Will produce an error, because the core of *Geo Meshing v3.2* does not accept spaces in folders and files path. In order to work, the previous folder should be renamed as:

*C:\Users\user1\Documents\testing_geo_meshing*

## 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 v3.2*.

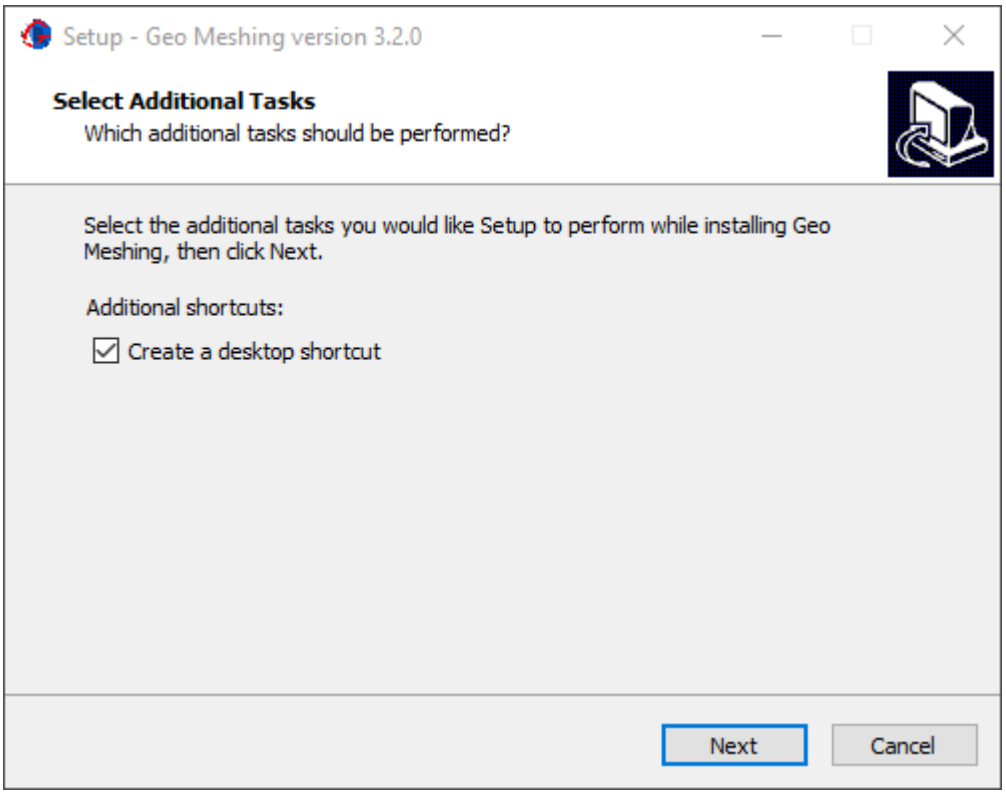


**Figure 1: *Geo Meshing v3.2* Installation Dialog Window.**

## 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 note that Geo Meshing v3.2 will not run properly if copied to a different source folder.*

This folder should contain the following files:

*- Geo Meshing v3.2.x.exe*

*- geo_me_3d.exe*

*- topo_render_v3.dll*

*- 2D_render.dll*

*- SCC_render.dll*

- *bmaplib.dll*

- *Microsoft.Maps.MapControl.WPF.dll*

- *GMManual3.0.0.pdf*

- *Resources [folder]*

- *MapData [folder]*

## 3.3. Registration

*Geo Meshing v3.2* needs the file "Geo_meshing.key" to run. Please go to *Geo Meshing v3.2* home page (*http://www.geomeshing.com)* and select the appropriate alternative under the license section. Once the appropriate license type has been added to the cart, please check out to obtain the license. Online license section still includes a free option.

The *Geo Meshing v3.2* code can be obtained under Help/About menu, by clicking the appropriate option (see Figure 2).

To avoid mistyped information, highlight *Geo Meshing v3.2* code and then right click on it, copy/paste into the website, when checking out your cart (see Figure 3). You will be able to download your license once the check out process is completed. This file needs to be uncompressed and stored in *C:\Program Files (x86)\Geo Meshing.*

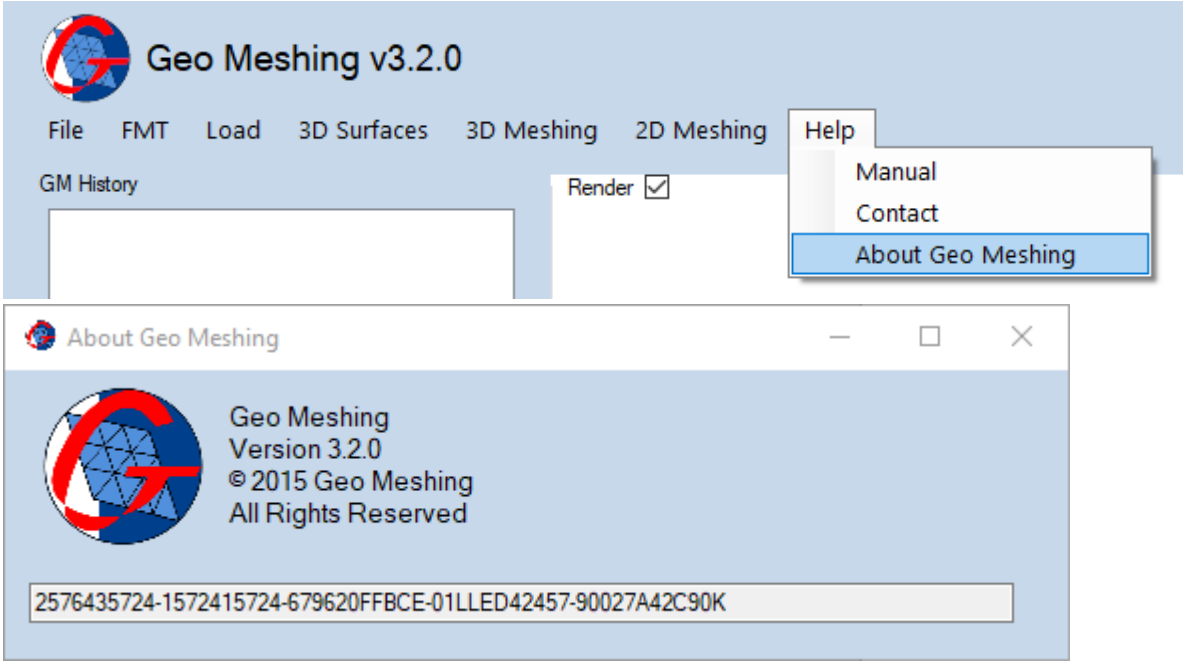


**Figure 2: *Geo Meshing v3.2* Code.**

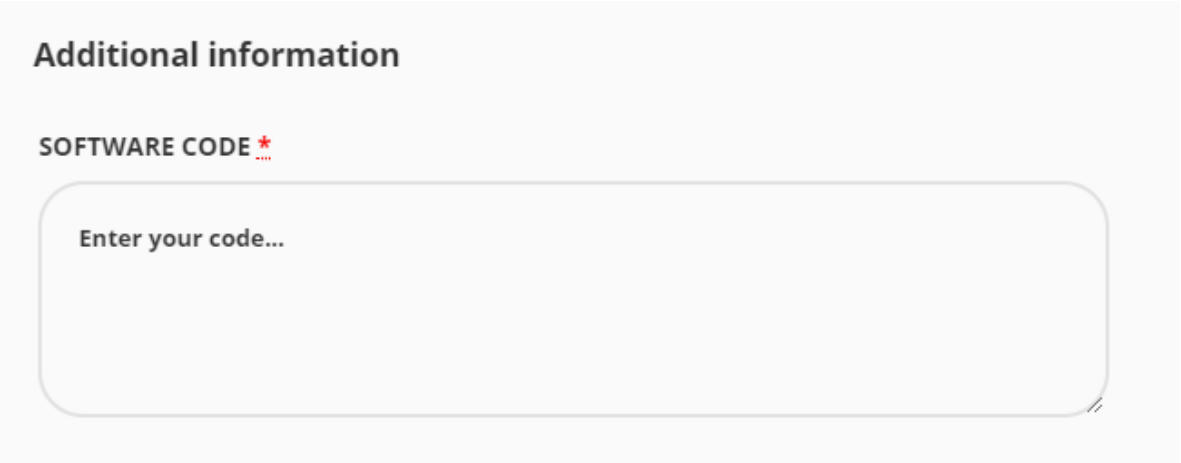


**Figure 3:** ***Geo Meshing v3.2 Webpage Paste Code.***

# 4. *Geo Meshing v3.2* Overview

Figure 4 shows the main components of *Geo Meshing v3.2*.

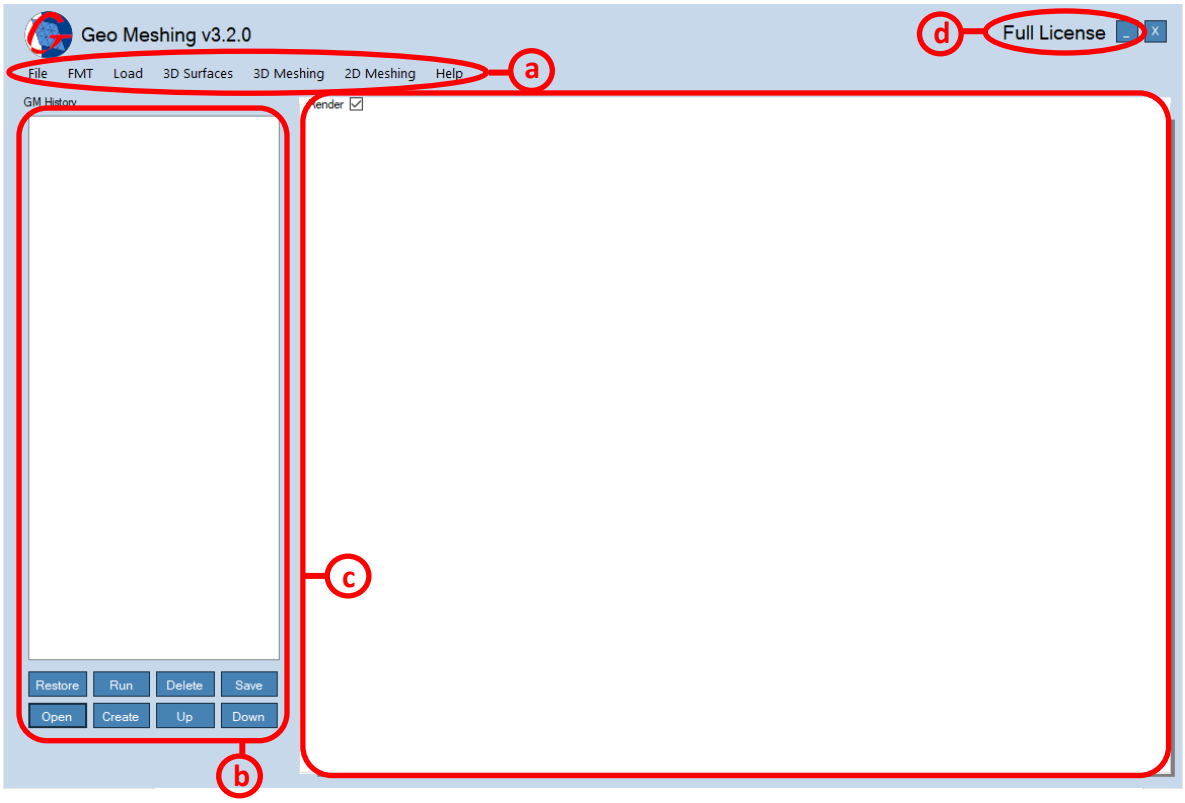


**Figure 4: *Geo Meshing v3.2* Overview.**

a) **Main Menu**: The strip menu area provides options for *Geo Meshing v3.2.* 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 v3.2* 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 v3.2* 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 v3.2*, it can be a "Test License" or it can be a "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 5.


**Figure 5: *History commands*.**

- **Restore:** This option restores the event selected in the *Geo Meshing v3.2* history windows. Before using this option, select an event in the history window. Alternatively, double click the event you want to restore.
- **Run:** This option runs a series of events defined by a range. Figure 6 shows the dialog windows for defining the range. *From* and *To* are the numbers shown in the *Geo Meshing v3.2* history windows. Before using this option, select an event in the history window.

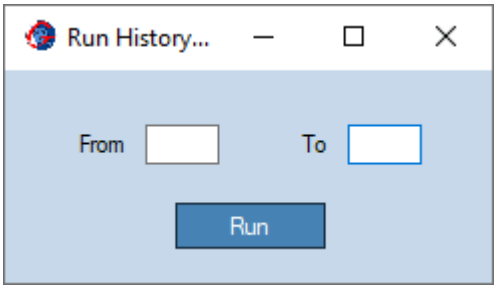


**Figure 6: Geo Meshing History – Run, define range dialog window.**

- **Delete:** This option deletes the event selected in the *Geo Meshing v3.2* history window. Before using this option, select an event in the history window.
- **Save:** This option saves the entire *Geo Meshing v3.2* history to the hard drive. If a file name for the *Geo Meshing v3.2* history has not been defined, one will be required during the saving process.
- **Open:** This option opens a *Geo Meshing v3.2* history. This option dialog box is described in section 5.1.1.
- **Create:** This option creates a *Geo Meshing v3.2* history file. This option dialog box is described in section 5.1.1.
- **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.

# 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.

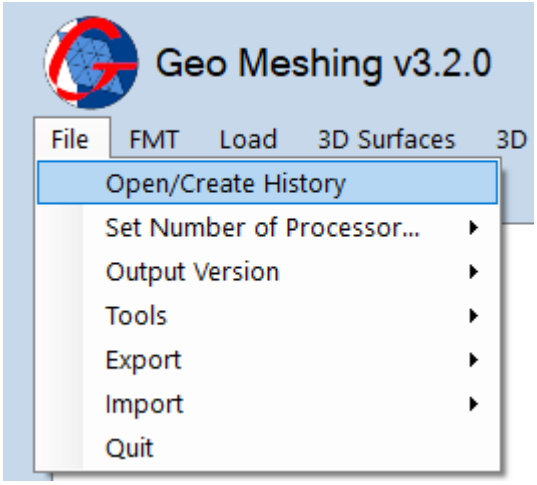


**Figure 7: File Menu – Open/Create History.**

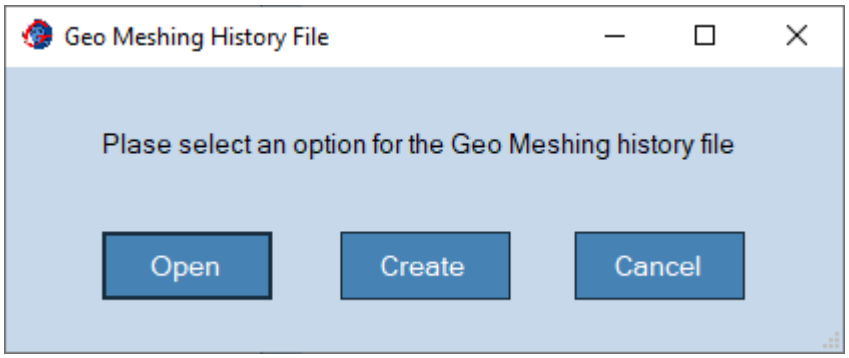


**Figure 8: Open/Create history dialog window.**

### 5.1.2. Set Number of Processor Menu

- **File Menu, Set number of processors**: This option (shown in Figure 9) has a submenu with four options.

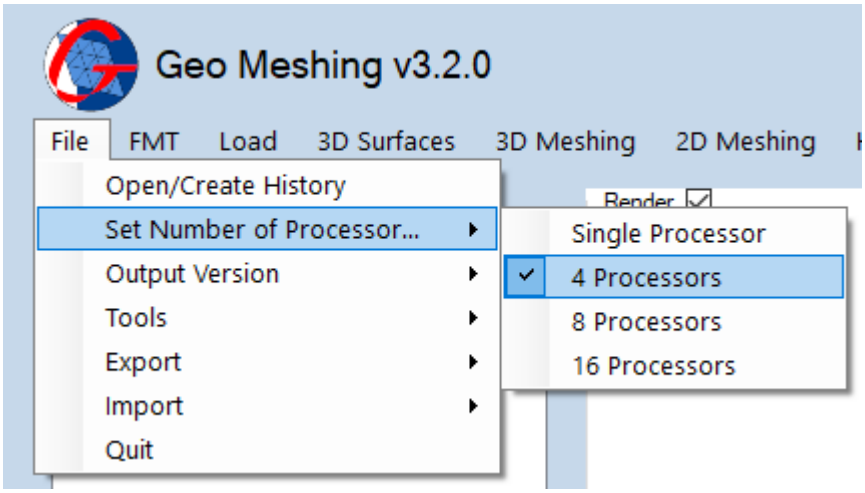


**Figure 9: File Menu – Set Number of Processors.**

- **Single Processor:** Select this option to use one processor to develop meshes.
- **4 Processors:** Select this option to use <u>four</u> processors to develop meshes.
- **8 Processors:** Select this option to use <u>eight</u> processors to develop meshes (not active with Test License).
- **16 Processors:** Select this option to use <u>sixteen</u> processors to develop meshes (not active with Test License).
  - **Hint:** The larger the number of processor 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.

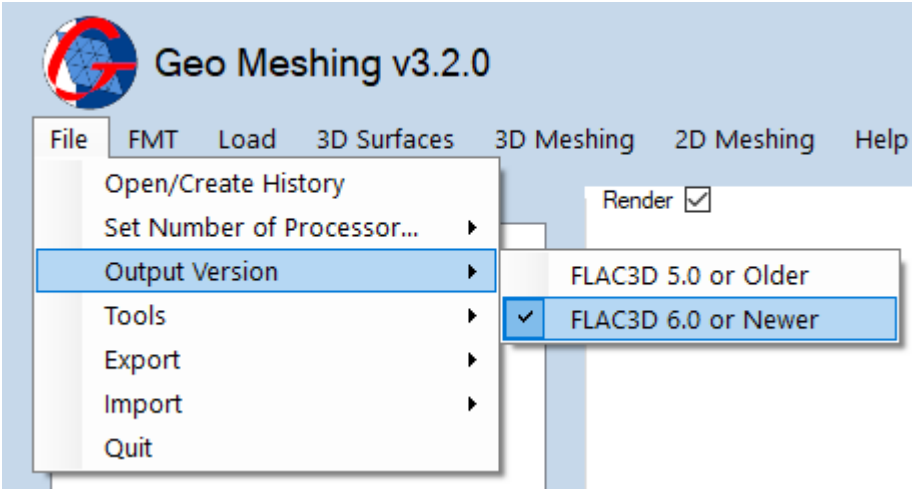


Figure 10: File Menu – Output Version.

  - **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. Tool

- **File Menu, Tool**: This option (shown in Figure 11) has a submenu for developing the stage capacity curve.

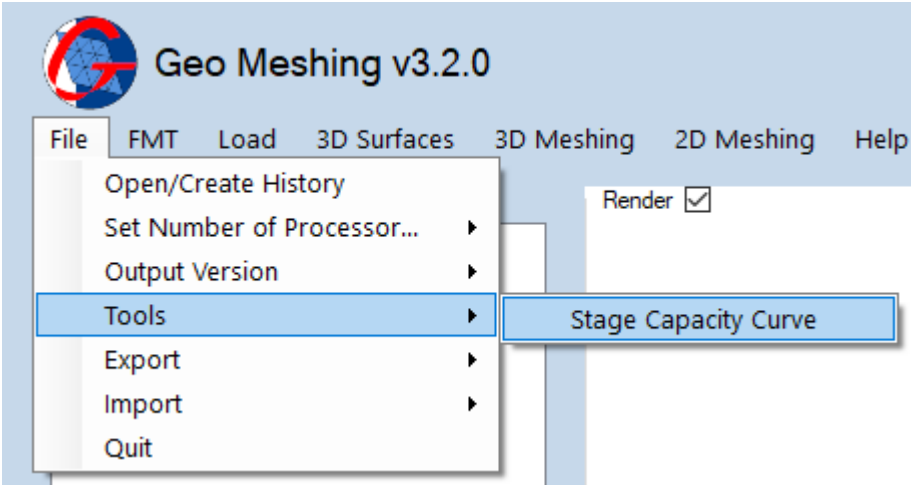


Figure 11: File Menu – Tool.

- **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 v3.2* file, with the 3D surface containing the higher elevation.
  - **Output Bot. Surface:** opens a *Geo Meshing v3.2* 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.

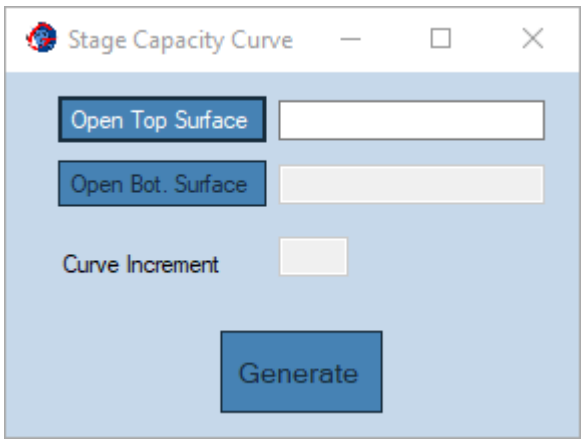


**Figure 12: Stage Capacity Curve dialog window.**

### 5.1.5. Export

- **File Menu, Export**: This option (shown in Figure 13) has a submenu with three options; *GM to CAD Script, GM to FLAC3D® Geometry and Selected Coordinates*.

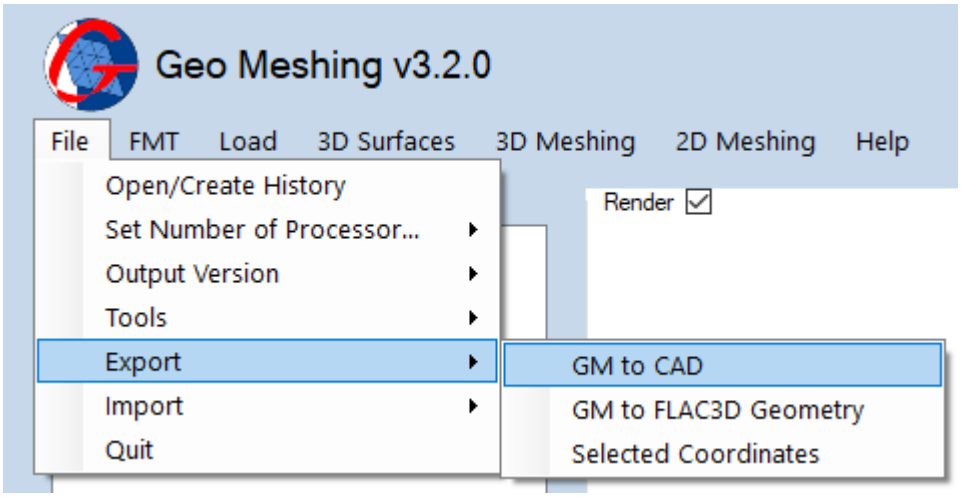


**Figure 13: File Menu – Export.**

- **GM to CAD Script:** Opens a dialog window (shown in Figure 14) for converting *Geo Meshing v3.2* 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 v3.2* 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).

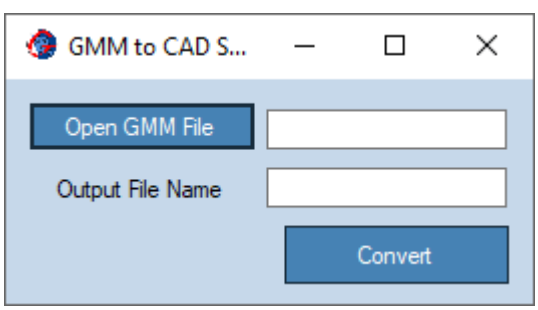


**Figure 14: *Geo Meshing v3.2* to CAD script surface dialog window.**

- **GM to Flac3D® Geometry:** Opens a dialog window (shown in Figure 15) for converting *Geo Meshing v3.2* 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 v3.2* 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.

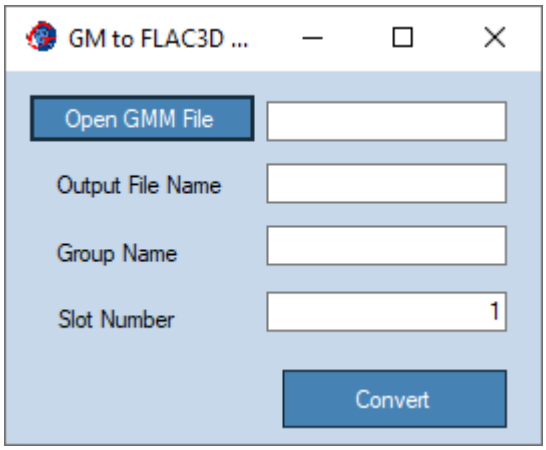


**Figure 15: *Geo Meshing v3.2* to Flac3D® Geometry dialog window.**

- **Selected Coordinates:** This option shows the dialog window of Figure 16, 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.

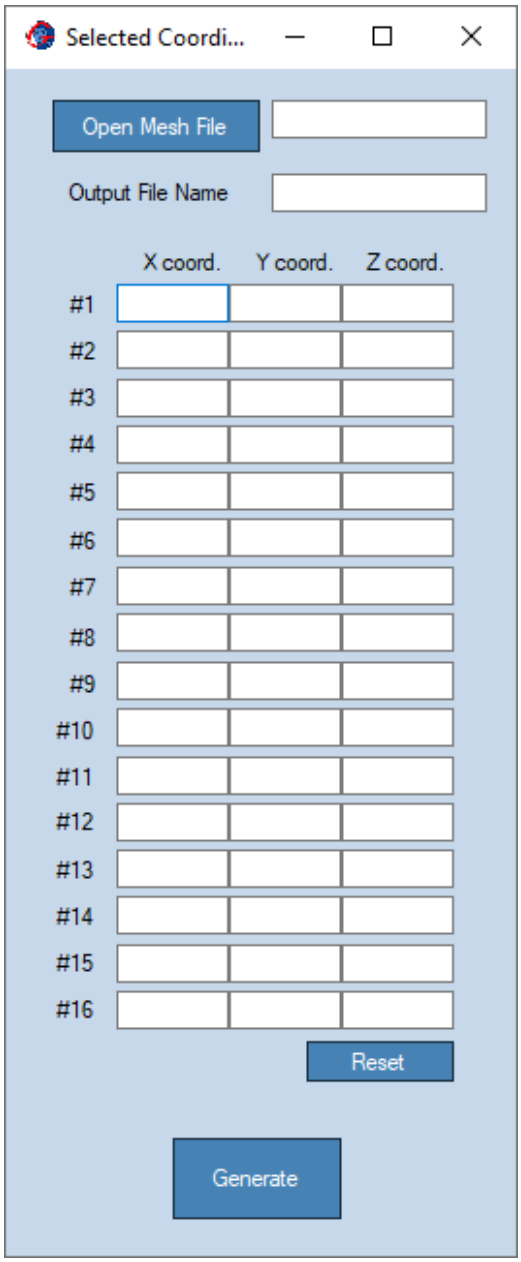


**Figure 16: File Menu – Selected Coordinates dialog window.**

### 5.1.6. Import

- **File Menu, Import**: This option (shown in Figure 17) has a submenu with *GID to GM* option.

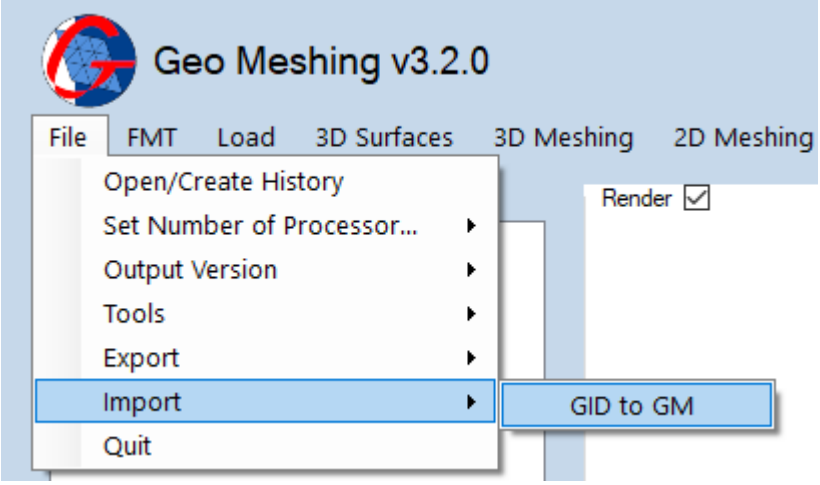


**Figure 17: File Menu – Export.**

  - **GID® to GM:** Opens a dialog window (shown in Figure 18) for converting GID® meshes files (*.msh) to *Geo Meshing v3.2* 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 v3.2* Mesh).
    - **Convert:** starts the process.

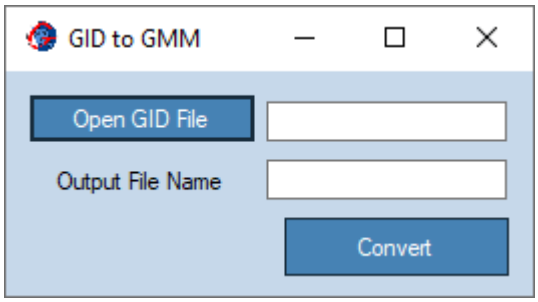


**Figure 18: GID® to *Geo Meshing v3.2* dialog windows.**

### 5.1.7. Quit

- **File Menu, Quit**: Exits from *Geo Meshing v3.2* (shown in Figure 19).

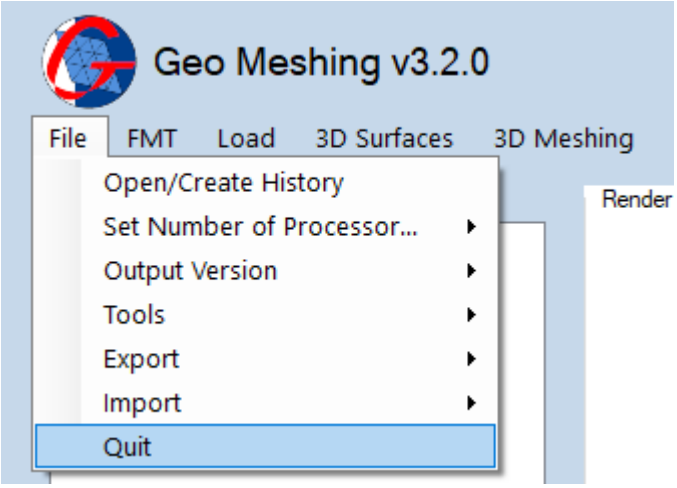


**Figure 19: File Menu – Quit.**

## 5.2. FMT (Flat Mesh & Topography)

This option opens the dialog window shown in Figure 20.

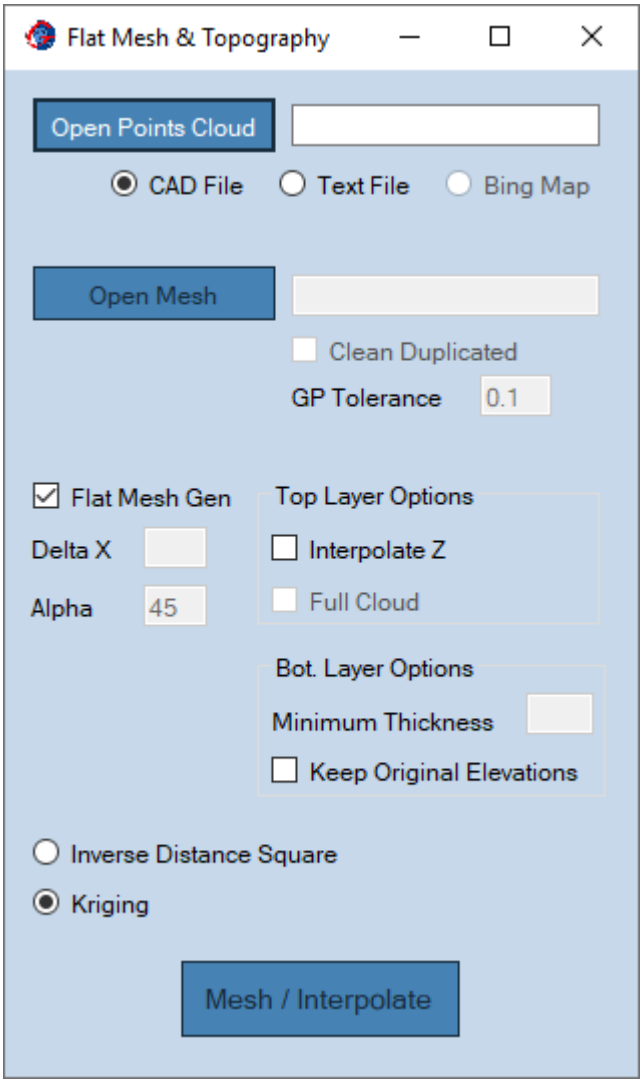


**Figure 20: FMT (Flat Mesh & Topography) Menu.**

- **Open Points Cloud**: This option helps selecting a cloud of point that represents a surface. Using this cloud of points a *Geo Meshing v3.2* 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 21 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 v3.2* will extract vertex information from all layers.
- **Select Layer**: Select the layer from which vertex are going to be imported in *Geo Meshing v3.2*, use the drop down 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.

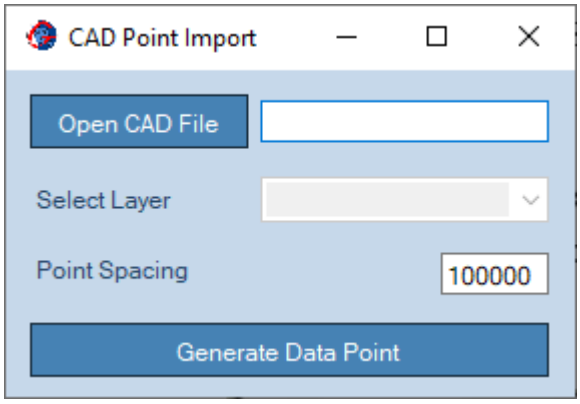


**Figure 21: CAD Point Import.**

- **Text File:** If this option is selected, opens a dialog window for importing a *.txt file containing the cloud of points representing a surface.
  **Note:** 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 v3.2* 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:

$$\left.\begin{array}{l} X1_1\ Y1_1\ Z1_1 \\ X2_1\ Y2_1\ Z2_1 \\ X3_1\ Y3_1\ Z3_1 \end{array}\right\} \text{Triangle 1}$$

$$\vdots$$

$$\left.\begin{array}{l} X1_n\ Y1_n\ Z1_n \\ X2_n\ Y2_n\ Z2_n \\ X3_n\ Y3_n\ Z3_n \end{array}\right\} \text{Triangle n}$$

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, 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 22.

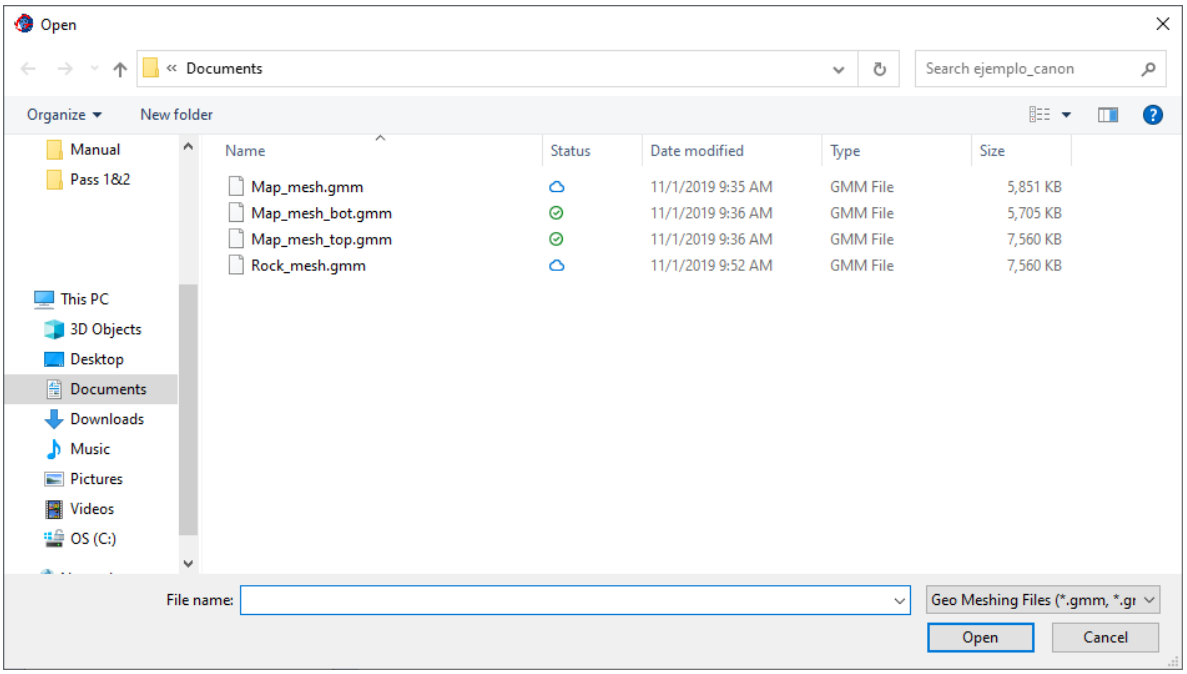


**Figure 22: Load Menu.**

- Navigate to the folder where you have your *Geo Meshing v3.2* 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 23) defines a Dam landform over any given surface.

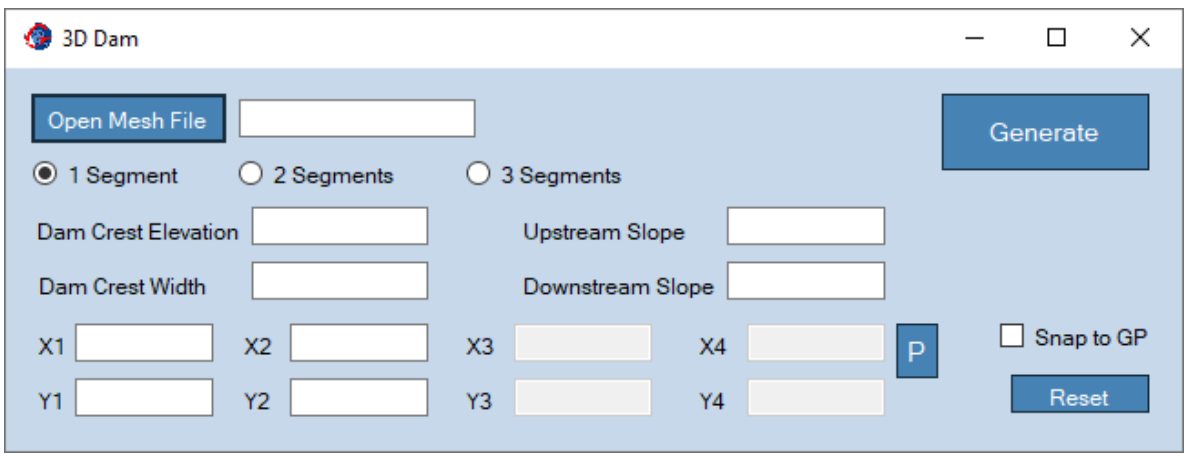


Figure 23: 3D Surfaces Menu – Dam.

- **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 24) defines a Circular Excavation landform over any given surface.

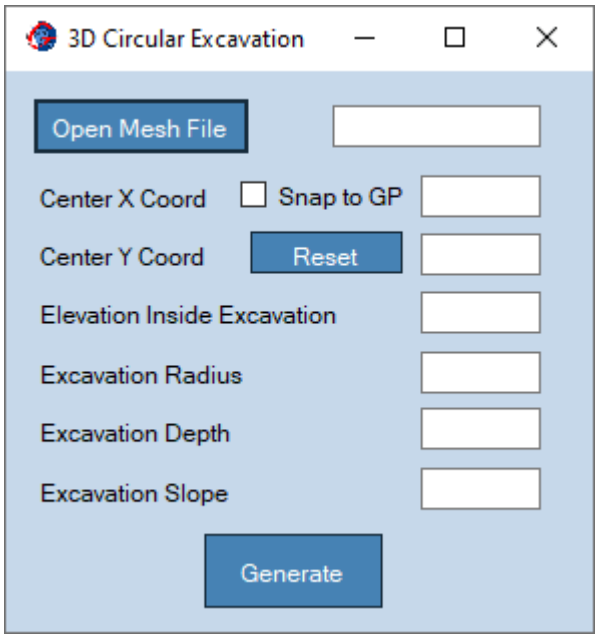


Figure 24: 3D Surfaces Menu – Circular Excavation.

- **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 25) defines a Vertical Fill (MSE Wall) landform over any given surface.

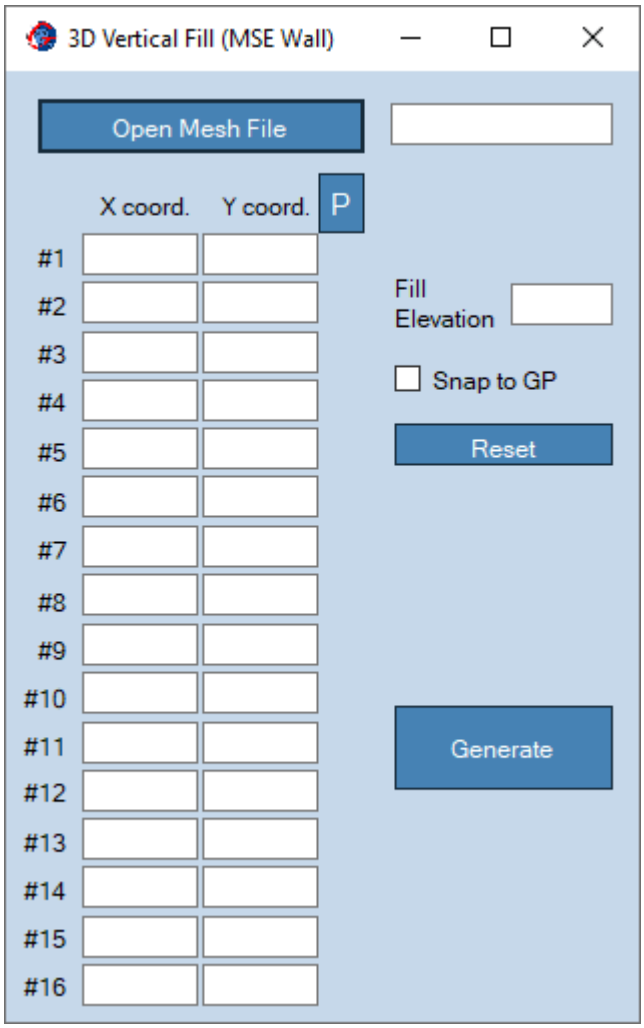


Figure 25: 3D Surfaces Menu – Vertical Fill (MSE Wall).

- **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 26) defines a Polygonal Fill landform over any given surface.

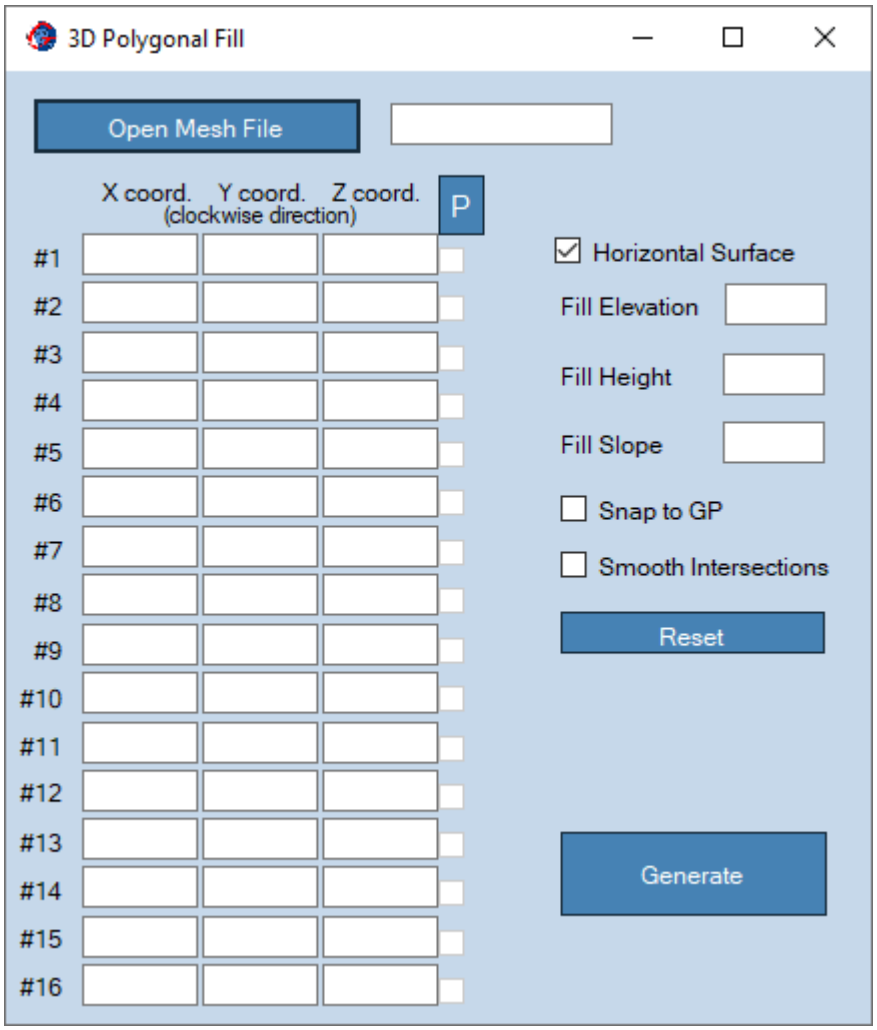


**Figure 26: 3D Surfaces Menu – Polygonal Fill.**

- **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 <u>must</u> 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 v3.2* 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 27) defines a Polygonal Excavation landform over any given surface.

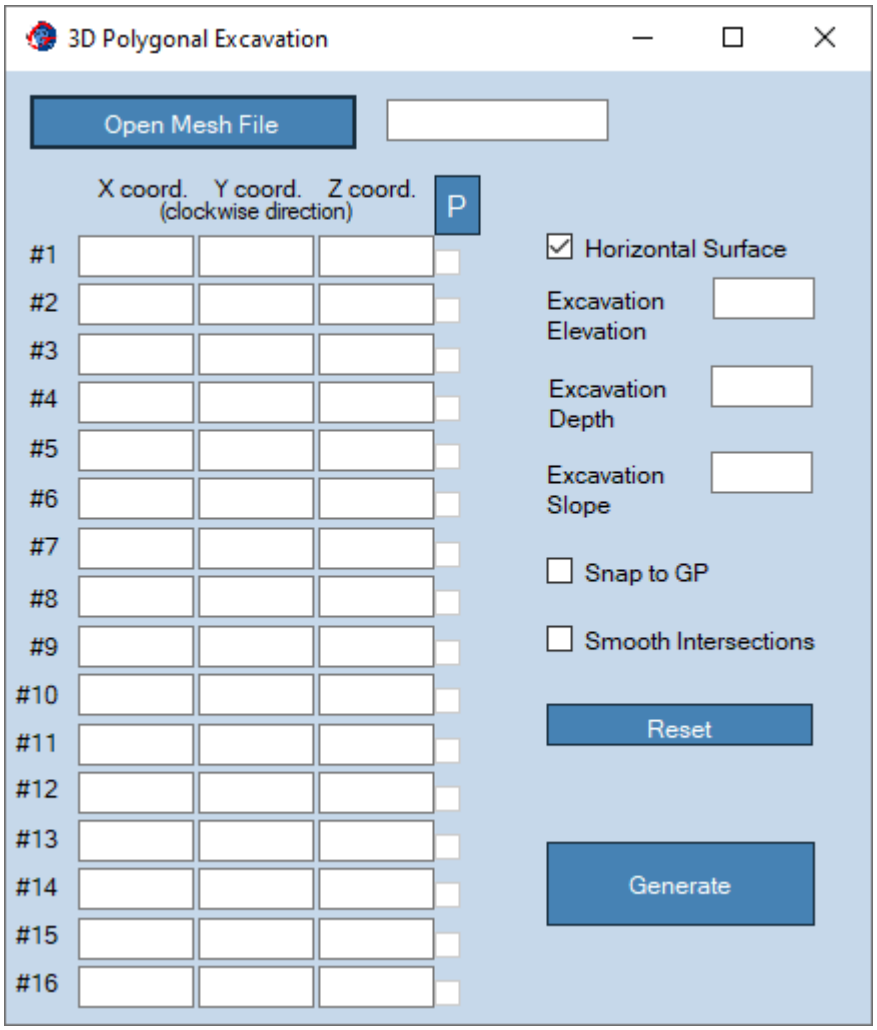


**Figure 27: 3D Surfaces Menu – Polygonal Excavation.**

- **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 <u>must</u> 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 v3.2* 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 28) 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.

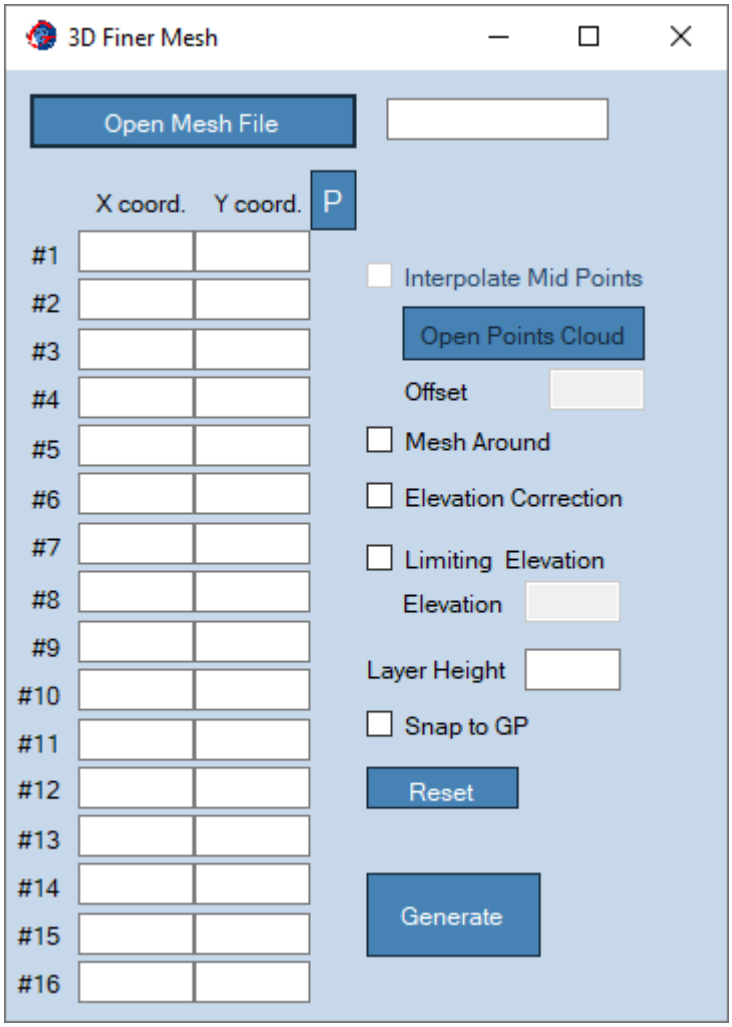


**Figure 28: 3D Surfaces Menu – Finer Mesh.**

- **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 v3.2* 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 29) is very similar to the previous one, but allows for horizonal 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.

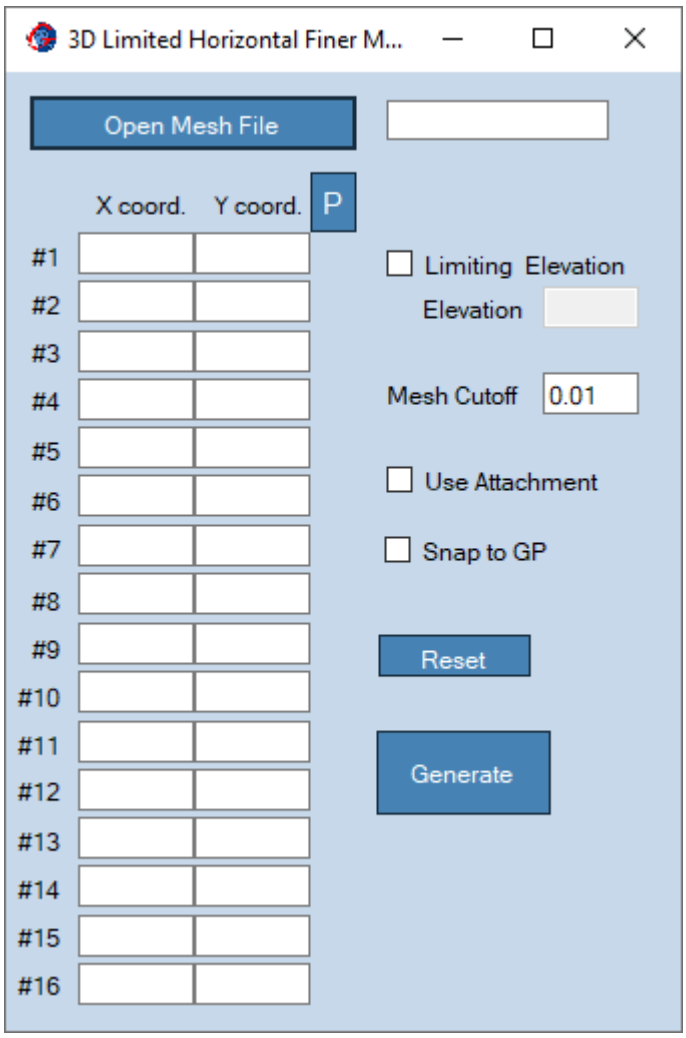


**Figure 29: 3D Surfaces Menu – Horizontal Finer Mesh.**

  - **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 <u>must</u> 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 30) produces a code for creating interfaces over an irregular surface created with *Geo Meshing v3.2*. This option does not require the execution of *3D Meshing* after generation.

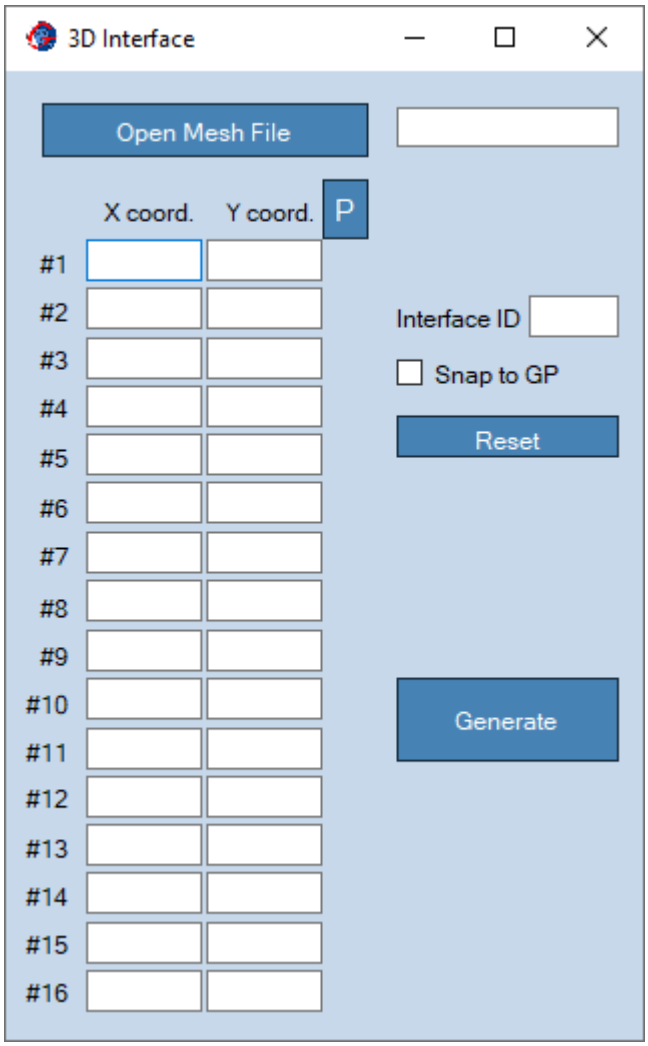


Figure 30: 3D Surfaces Menu – Interface Mesh

- **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 31) defines a Wedge Excavation landform over any given surface.

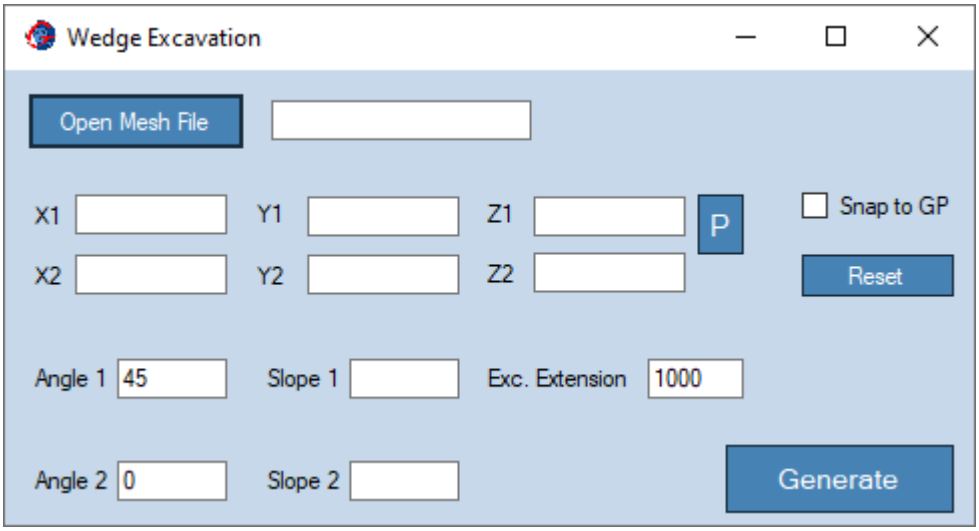


Figure 31: 3D Surfaces Menu – Wedge Excavation

- **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 32) operates an Addition or Subtraction between two 3D surfaces.

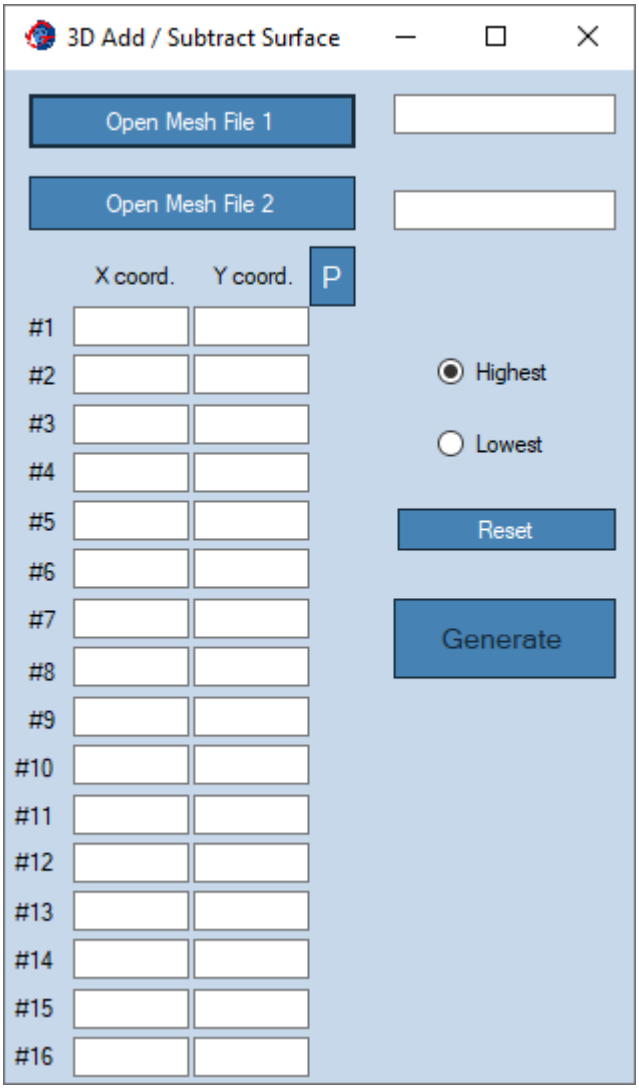


Figure 32: 3D Surfaces Menu – Add / Substract

- **Open Mesh File 1**: Opens the *.gmm file on which the Addition or Subtraction process is going to be performed.

- **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 33.

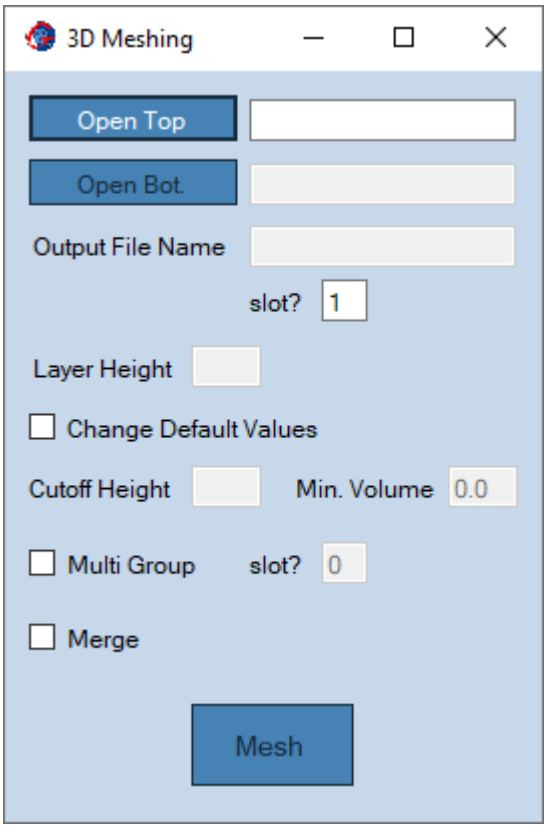


Figure 33: 3D Meshing Section

- **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 other meshing processes.

- **Layer Height:** This is the height of the elements filling the volume between the top and bottom surfaces.

- **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***.**
    - **Min. Volume**: Sets volume threshold, thus elements with a volume less than *Min Volume* are not created.

- **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.

- **Merge**: This option merges 2 opposite elements to form one element. This option considerably reduces the number of elements or zones, but keeps the number of GP (gridpoints).

- **Mesh**: Starts the 3D Meshing process.

## 5.6. 2D Meshing (not active with *Test License*)

This option executes *2D Meshing* section, as shown in Figure 34. This option is useful when big meshes are intended to be generated for FLAC®, and it consider ratios is several part of it.

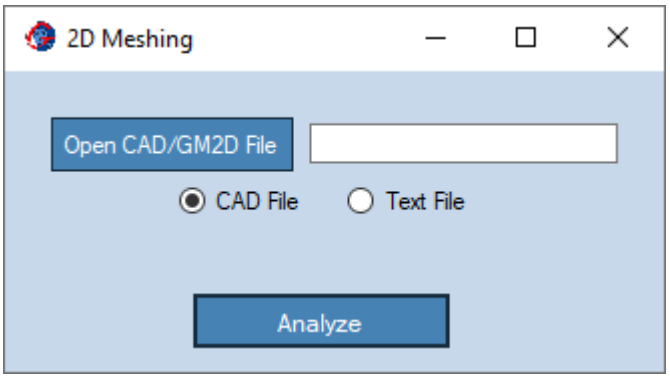


**Figure 34: 2D Meshing Dialog Window**

- **Open CAD/GM2D File**: Opens the *.DXF or *.gm2d file containing the 2D geometry. The type of file that this dialog windows opens, depend on whether *CAD File* or *Text File* is selected.

  The CAD file must have at least 2 layers; one named *Main* and the other named *Domain*. The rest of the layer are ignored. Both layers should have only closed polylines elements. *Main* should contain the geometry where the mesh needs to be finer. *Domain* should contain the overall domain of the model. Once they are analyzed, *Main* and *Domain* layers are displayed in red and blue, respectively, as shown in Figure 35.

  - **Warning:** The CAD file (*.DXF ) MUST be saved in R12 version, or data won´t be read.

  The text file should have the following format:

  1. MAIN or Domain key word
  2. Number of X,Y pairs. For example; a triangle has 4 X,Y pairs, since the first one has to be repeated.
  3. X,Y pairs

  The following is an example of input text file, and it corresponds to the example of Figure 35:

  MAIN 5 300 200 361.342 238.330 386.342 238.3309 447.684 200 300 200
  MAIN 7 361.80447992 238.11028177 … … 361.80447992 238.11028177
  MAIN 8 410.12574214 269.20339703 … … 410.12574214 269.20339703
  DOMAIN 7 0 200 … … 0 200

DOMAIN 20 0 130 … … 0 130

DOMAIN 18 0 0 … … 0 0

- **Analyze**: Runs the analysis of either the CAD or Text file, and display the image shows in Figure 35.

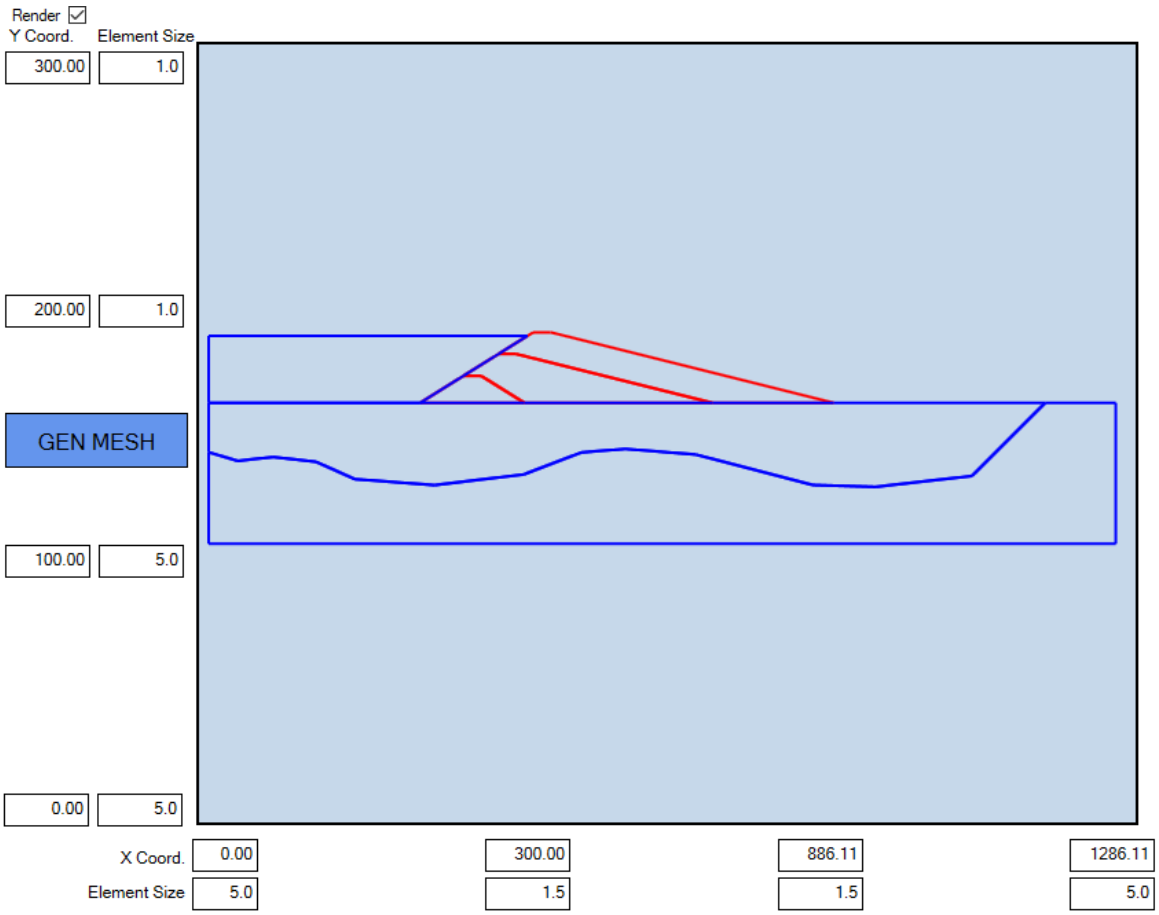


**Figure 35: 2D Meshing Geometry in Render Area**

The render area of *Geo Meshing v3.2* shows the geometry along with 16 boxes containing X,Y coordinates and elements sizes. The vertical (or Y) direction, shows 4 boxes for coordinates and 4 boxes for element sizes. Likewise, the horizontal (or X) direction also shows 8 boxes. The coordinates are automatically obtained from the CAD or Text File.

In order to understand how these coordinates are obtained, one can visualize the entire geometry divided in 9 squares, as shown in green in Figure 36. Generally, internal coordinates are defined by the extension of MAIN polylines, while overall geometry is defined by the DOMAIN extension. The position of the fine mesh (red lines) is defined in the CAD or Text File, as well where this zone is located within the 9 squares and how many of them uses.

The element sizes are defined so a small element of 1 by 1.5 units is at the finer red area. At the boundaries an element of 5 units is selected. Transition are made in the code by using *ratios*. In the example of Figure 36, the domain is subdivided as follow:

In the vertical direction

- From elevation 0 to 100 units a constant height element of 5 unit is used
- From elevation 100 to 200 units a variable height element from 5 units to 1 unit is used
- From elevation 200 to 300 units a constant height element of 1 unit is used

In the horizontal direction

- From 0 to 300 units a variable length element from 5 units to 1.5 unit is used
- From 300 to 886.11 units a constant length element of 1.5 unit is used
- From 886.11 to 1286.11 units a variable length element from 1.5 units to 5 unit is used

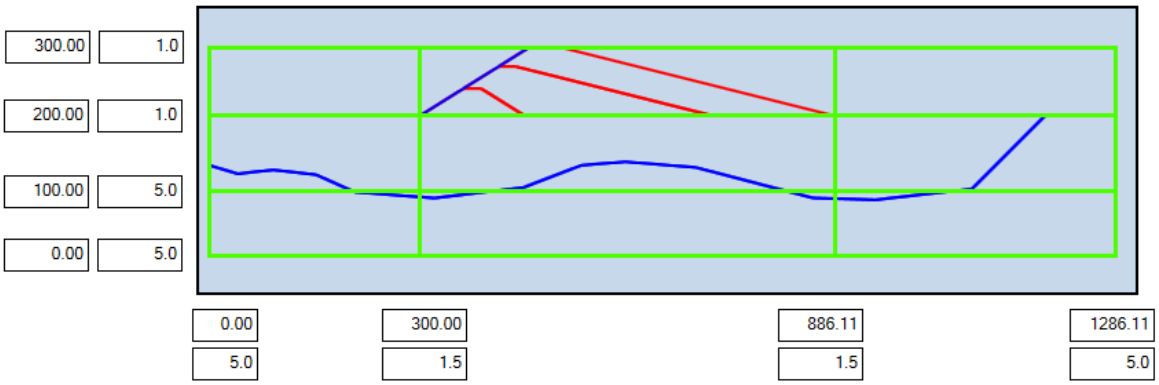


**Figure 36: 2D Geometry Domain – 9 Square Division**

The coordinates and element sizes can be edit, if one needs further adjustments. Once the number in all 16 boxes are in agreement, click the *GEN MESH* button (blue button) of Figure 35, for generating the FLAC® Mesh.

## 5.7. Help Menu

### 5.7.1. Manual Option

- **Help Menu, Manual**: This option (shown in Figure 37) opens this manual.

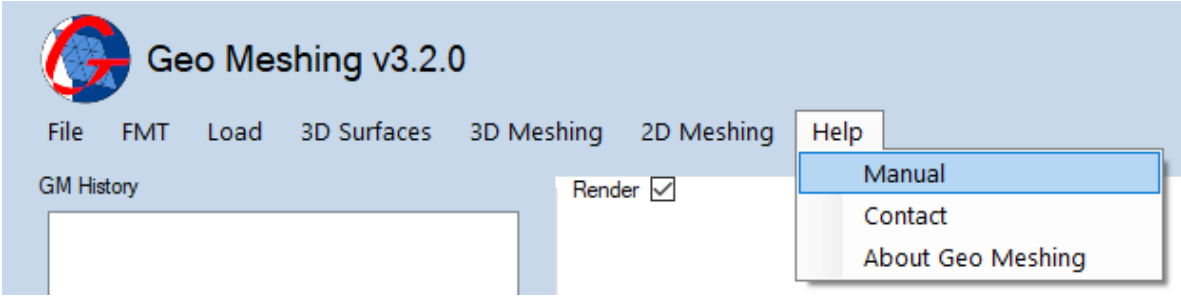


**Figure 37: Help Strip Menu – Manual.**

### 5.7.2. Contact Option

- **Help Menu, Contact**: This option (shown in Figure 38) displays support information.

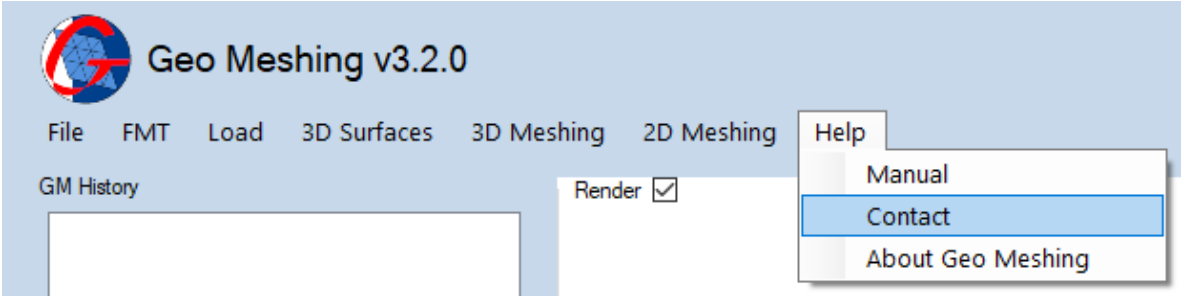


**Figure 38: Help Strip Menu – Contact.**

### 5.7.3. About Geo Meshing Option

- **Help Menu, About Geo Meshing**: This option (shown in Figure 39) displays *Geo Meshing v3.2* information along with the *Geo Meshing v3.2* code (See section 3.3 for more information).

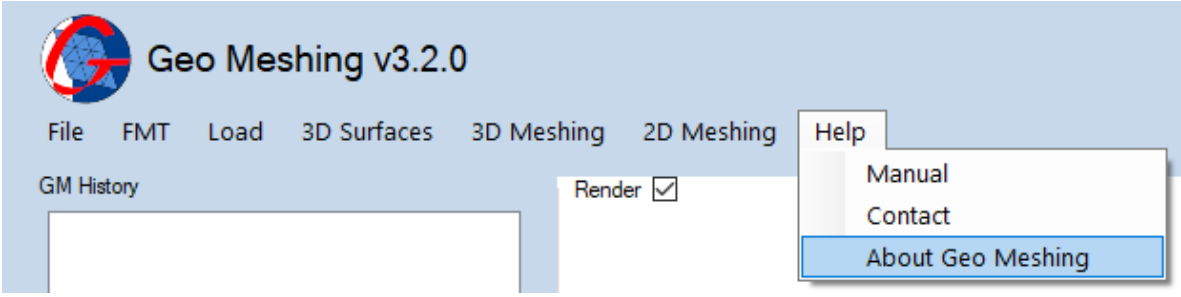


**Figure 39: Help Strip Menu – About Geo Meshing.**

# 6. Render

The *Render* section is a built-in window in *Geo Meshing v3.2*. 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 40: 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. 2D Render Controls

There are no controls in the 2D mode, but changes to the domain parameters can be made as explained in section 5.6.

## 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 v3.2* (*.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 v3.2*. In addition, the output generated by *Geo Meshing v3.2* 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 v3.2* will generate a file called *finer_source.dat (hfiner_source.dat)*, and for the second refinement *Geo Meshing v3.2* 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 v3.2* code.
  - You may need administrator privileges to run the program. If you have administrator privileges, please configure *Geo Meshing v3.2* 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 v3.2* 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. See section 3.1.

- 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.