

TETRIUM

(User Manual)

Unstructured mesh generator for geophysical modelling and inversions

Version: 1.01

Last updated: January 2026

Copyright © 2025-2026 Geotexera Inc.

TETRIUM is a software package that integrates multiple programs to generate unstructured meshes for geophysical modelling and inversions. It is compatible with Windows, macOS, and Linux, and is provided without any warranty. If you use TETRIUM in your work, please acknowledge it by providing an appropriate citation.

CONTENTS

Introduction.....	2
General Structure.....	3
Basic Settings.....	6
Regions.....	12
Surface Refinement.....	16
Volume Refinement.....	21
Mesh Operators.....	26
Output Settings.....	31
Important Notes.....	34
Examples.....	36

1. INTRODUCTION

In TETRIUM, the meshing engine is Gmsh, a finite-element mesh generator whose source code and pre-compiled binaries are available at <https://gmsh.info>. Note that you need to download the *Software Development Kit (SDK)* for your operating system. TETRIUM interfaces with Gmsh solely as an external program. TETRIUM does not include, distribute, or modify the Gmsh source code or binaries. Users are required to obtain Gmsh independently from its official website and are responsible for complying with the terms of the GNU General Public License (GPL) or any other license provided by the Gmsh authors.

The integration between TETRIUM and Gmsh is limited to external invocation and does not involve linking or embedding Gmsh code within TETRIUM. Accordingly, the GPL governing Gmsh does not apply to TETRIUM's source code. All obligations related to the use, distribution, or licensing of Gmsh rest with the user.

For information about alternative licensing options, users may consult the Gmsh documentation or contact the Gmsh authors directly.

2. GENERAL STRUCTURE

The following tutorials introduce TETRIUM's features, beginning with the general structure of the software.

The menu bar contains three main menus:

- **File Menu**
 - Load Settings (Ctrl+O)**
 - Loads a previously saved '.ttrm' settings file
 - Restores all input parameters from the saved configuration
 - Save Settings (*.ttrm) (Ctrl+S)**
 - Saves the current configuration to a '.ttrm' file
 - Includes all input parameters for later reuse
 - Exit (Ctrl+E)**
 - Closes the application
 - Prompts for confirmation before closing
- **View Menu**
 - Log (Ctrl+L)**
 - Toggle to show/hide the Execution Log section at the bottom
 - Unchecked by default (log hidden)
 - When visible, shows real-time execution progress and messages
- **Help Menu**
 - About**
 - Displays application information (currently shows placeholder message)
 - Video Tutorials**
 - Access to video tutorials (currently shows placeholder message)
 - Documentation**
 - Access to documentation (currently shows placeholder message)

The main interface contains six tabs:

- **Basic Settings**
 - TOPOGRAPHY Section:**
 - Choose "Fixed" (elevation value) or "From File" (topography file)
 - Topography file input and decimation options
 - REGION OUTLINE:** Button to open the Region Outline Window for visualizing and marking region boundaries
 - GMSH Path:** Path to the Gmsh software (SDK files)
 - Mesh Algorithm Settings: 2D Algorithm, 3D Algorithm, Mesh Optimize, Mesh Optimize Threshold, Mesh Smoothing
- **Regions**
 - COI (Core) Region:** Define as "Box" (coordinates) or "From File"
 - Sub-COI Region:** Sub-region with depth parameters
 - ROI (Inner Padding) Region:** Define as "Box" or "From File"



- POI (Outer Padding) Region:** Define as “Box” or “From File”
Air Region: Air region modeling with boundary refinement options
- Surface Refinement**
 - Refinement Around Stations:** On-surface station refinement
 - Refinement Along Lines/Loops:** On-surface line/loop refinement
 - Both include options for extra refinement nodes (Above in Air, below in COI, or None)
 - Decimate:** Options for decimating extra nodes on surface
- Volume Refinement**
 - Refinement Around Stations:** Volume station refinement for COI, ROI, and Air regions
 - Refinement Along Lines/Loops:** Volume line/loop refinement for COI, ROI, and Air regions
 - Refinement Of Zones:** Zone-based refinement
 - Decimate:** Options for decimating extra nodes in volume
- Mesh Operators**
 - Mesh Operator Type:** Choose “Standard” or “Boolean”
 - Import Geometries:** Up to 10 geometry files can be imported
 - Each file can be specified as “Points” or “Surfaces/Volumes” type
 - Used for boolean operations or additional geometry inclusion
- Output Settings**
 - Output Type:** Select output format (0, 2, 3, 5, or .msh)
 - Boundary Marker Value:** Value for boundary markers in output files
 - Root Name For Output Files:** Base name for generated mesh files
 - Output Location:** Directory where output files will be saved
 - Keep Intermediate Output Files:** Checkbox to preserve intermediate files (unchecked by default)
 - Post-Run:** Checkbox to start execution from STAGE 11 (for re-running with modified parameters)

Additional UI Elements are:

- **Run Button:** Starts the mesh generation process
- **Stop Button:** Stops the current execution (enabled during execution)
- **Execution Log:** Scrollable log window showing:
 - Current stage and run information
 - Progress messages
 - Success/error indicators
 - File operations and commands executed

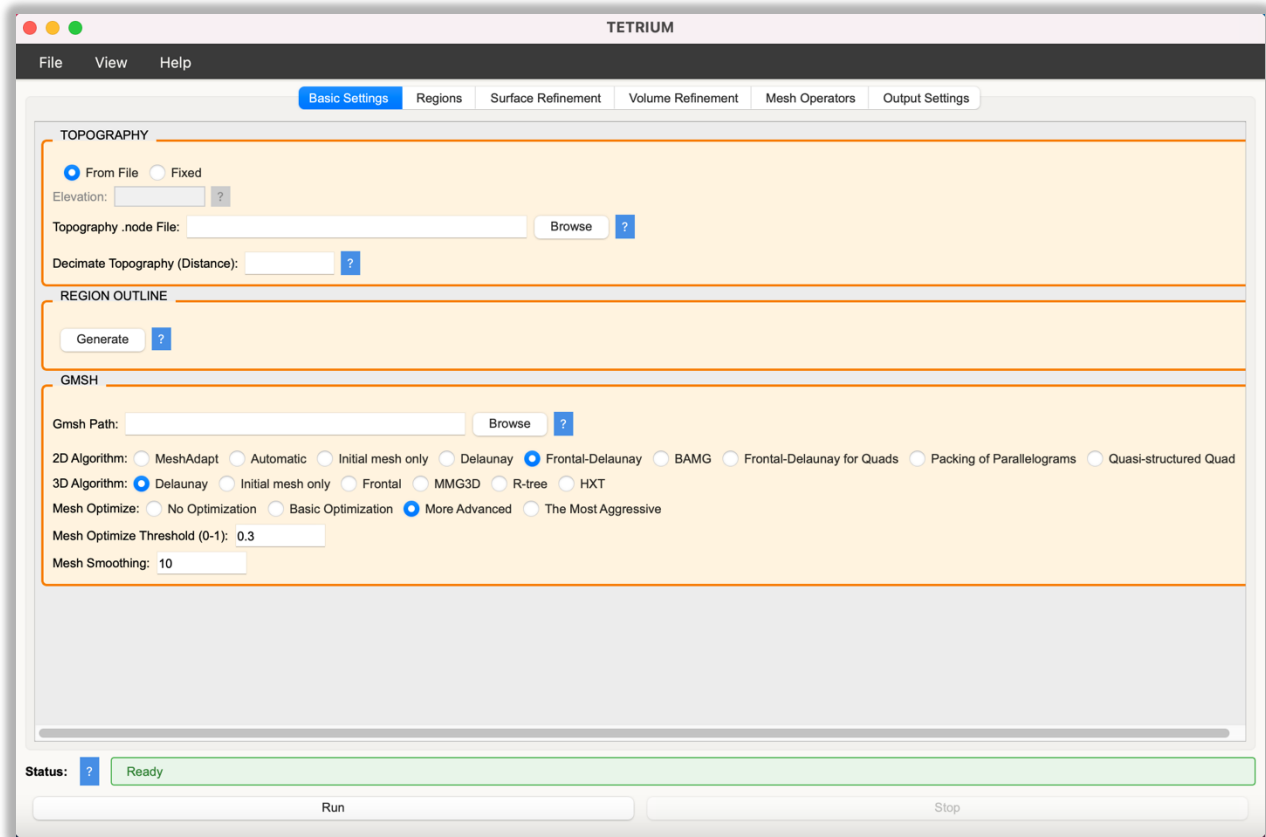


Figure 1: Software GUI.

3. BASIC SETTINGS

3.1. Topography

The Topography section in the Basic Settings tab defines how the surface elevation is handled in the mesh. It supports two modes: “Fixed” (constant elevation) and “From File” (read from a ‘.node’ file). The choice determines which processing stages run, and which inputs are required.

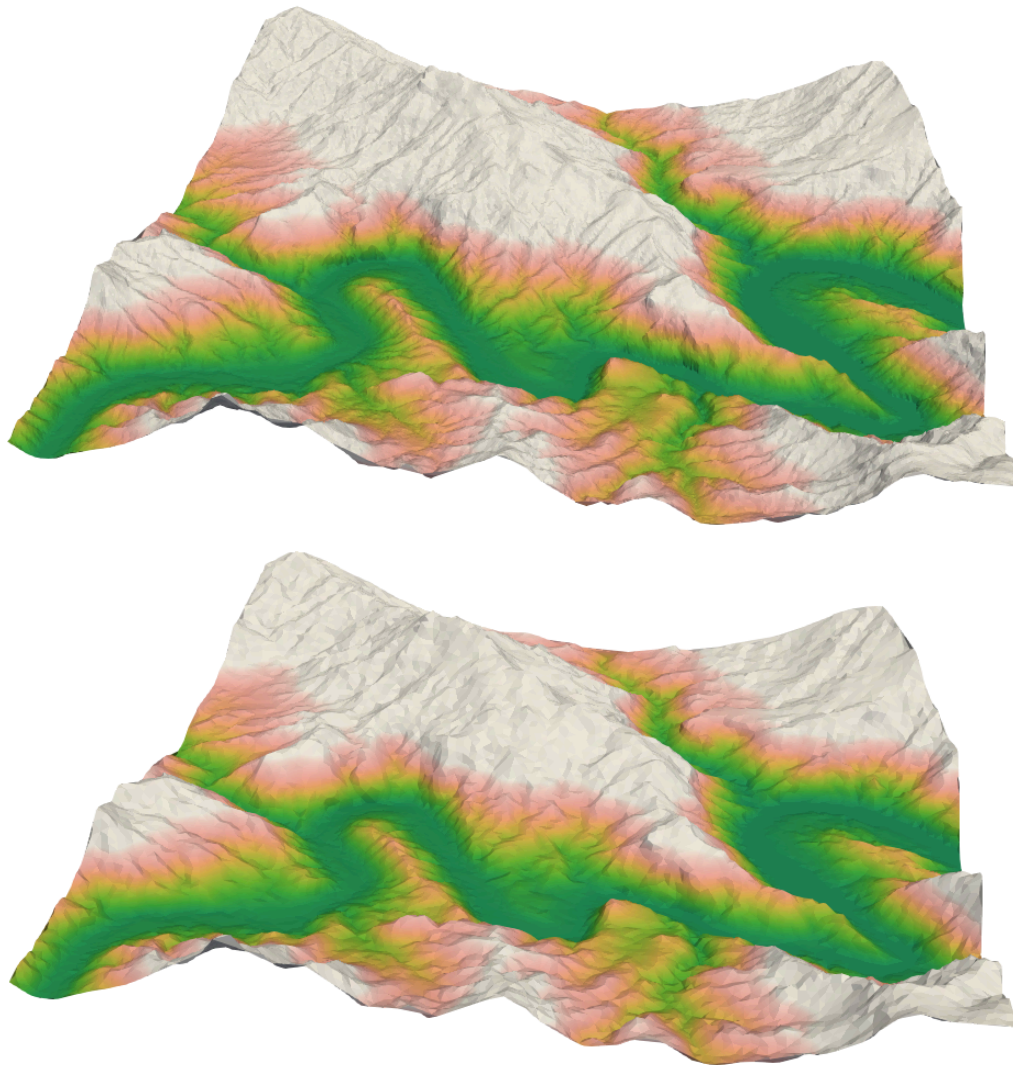


Figure 2: Topography before (top; 70k points) and after (bottom; 10k points) decimating.

Topography Type Selection

Two radio buttons control the mode:

“From File” (default): Reads elevation data from a ‘.node’ file. The file should contain surface elevation data in standard ‘.node’ format with a header line indicating the number of nodes and dimensions, followed by node data with coordinates (node_index x y z). This mode enables STAGE 5 (Topography Preparation) and STAGE 6 (Topography Interpolation), which process and integrate the topography into the mesh. Use the “Browse” button to select the file.

“Fixed”: Uses a constant elevation value across the domain. When selected, the “Elevation” field becomes active, and you enter a single numeric value. The topography file and decimation options are disabled. STAGE 5 and STAGE 6 are skipped, and the mesh uses the fixed elevation. If “Fixed” is chosen, the Sub-COI region option is ignored, as variable topography is required for Sub-COI depth calculations.

Decimate Topography Option (From File Mode)

The “Decimate Topography (Distance)” field is available when “From File” is selected. It removes nodes that are too close based on a maximum separation distance. For any two nodes separated by less than the specified distance, the second node is removed. This reduces file size and processing time. If not provided, the software defaults to 1.0E-6. The decimation runs in STAGE 5 before the topography is processed by GMSH.

3.2. Region Outline

The Region Outline section in the Basic Settings tab provides a tool for creating region boundary files. It opens a visualization window where you can view station or geometry files and mark points to define region outlines. These outlines can be saved and used as region files in the Regions tab.

Purpose and Functionality

The Region Outline tool helps create irregular region boundaries when simple box coordinates are insufficient. It visualizes VTU or ‘.node’ files in 2D (X-Y projection) and lets you mark points to form closed outlines. The marked points are saved to a text file with Z-parameters (Z-min, Z-max, Z-points) and can be used as the “COI (Core) Region File”, “ROI (Inner Padding) Region File”, or “POI (Outer Padding) Region File” in the Regions tab.

The Generate Button

Clicking the “Generate” button opens the Region Outline Window, a separate dialog for visualization and point marking. The window includes a 2D plot, file loading controls, point marking tools, and navigation options.

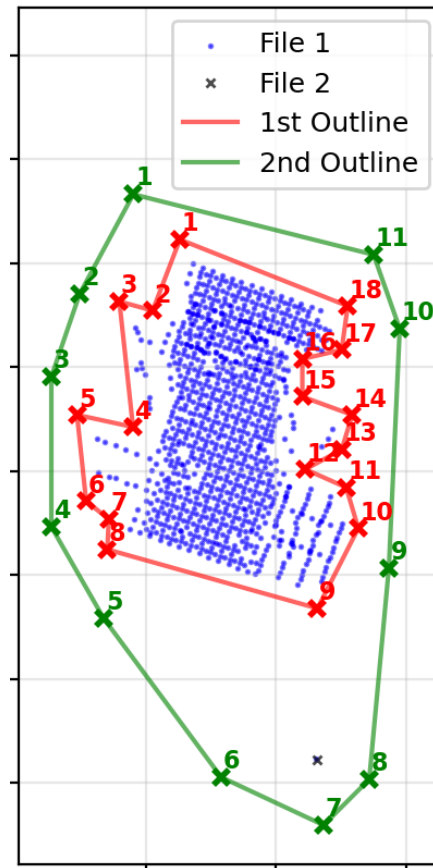


Figure 3: COI (red) and ROI (green) region outlines around stations (blue and black).

The Region Outline Window

The window supports loading up to three VTU or '.node' files simultaneously for comparison. File 1 appears as blue dots, File 2 as black crosses (x), and File 3 as red triangles. Files can be loaded in any order and replaced by clicking the same button again.

Creating Outlines

You can create up to three separate outlines (1st, 2nd, and 3rd), each displayed in a different color: 1st outline in red, 2nd outline in green, and 3rd outline in blue. To create an outline, click "Mark Points" to enable marking mode, then click on the plot to place points. The points are connected to form a closed outline. Click "Stop Marking" to save the current outline and move to the next. Note that the last connection between the last point and the first point will be made after clicking on stop marking. After the 3rd outline is created, the "Mark Points" button is disabled.

Point Management

All marked points from all outlines are listed in a scrollable list at the bottom of the window, showing the outline name, point number, and X and Y coordinates. You can select any point from any outline in the list and use the “Edit Points” button to modify its X and Y coordinates, or use “Delete Points” to remove it. The “Clear Points” button removes all marked points from all outlines.

Navigation and Visualization

The window includes zoom and pan controls. The “Zoom In” and “Zoom Out” buttons adjust the view, and you can also use the mouse wheel to zoom at the cursor position. The “Reset Zoom” button returns to the original view. You can pan by clicking and dragging with the left mouse button (when not in marking mode) or with the middle or right mouse button at any time. These controls help navigate large datasets and mark points precisely.

Saving Outlines

The “Save Points” button saves a selected outline to a text file. You choose which outline to save (1st, 2nd, or 3rd), then enter Z-min, Z-max, and Z-points values in a dialog. Z-min is the minimum Z coordinate for the region block (base/bottom), Z-max is the maximum Z coordinate (top, used when an Air Region is included and must be above topography), and Z-points is the Z coordinate assigned to all marked points (currently not used in this version, keep as 0.0). The saved file format includes a header line with Z-min and Z-max, followed by each point's X, Y, and Z-points coordinates.

Integration with Main Application

The saved outline files can be used directly in the Regions tab as region files. When you select “From File” for a region (COI, ROI, or POI) and browse for a file, you can select the text file saved from the Region Outline Window. This allows you to define irregular region boundaries based on station locations or other geometric features visualized in the window.

Use Cases

This tool is useful when you need region boundaries that follow station distributions, geological features, or other irregular patterns visible in your data files. Instead of manually creating coordinate files, you can visualize your data, mark points around the desired area, and generate a region file that follows the actual distribution of your stations or features. This is particularly valuable for geophysical applications where region boundaries should align with survey layouts or geological boundaries.

3.3. GMSH

The GMSH section in the Basic Settings tab configures the Gmsh mesh generator. It includes the GMSH installation path and mesh generation parameters (algorithms, optimization, and

smoothing). These settings are written into the generated '.geo' files and control how GMSH creates the mesh.

GMSH Path

The "GMSH Path" field specifies the folder containing the Gmsh files (i.e., *Gmsh Software Development Kit files; SDK*). Use the "Browse" button to select the unzipped directory. The software uses this path to run Gmsh commands during mesh generation, particularly in STAGE 3 (for initial topography surfaces) and STAGE 11 (for the final 3D mesh). The path should point to the Gmsh folder that contains the 'bin', 'include', and 'lib' folders. This is a required field; if not specified, the mesh generation process cannot proceed.

2D Algorithm Selection

The "2D Algorithm" option selects the algorithm for 2D surface meshing. Nine options are available, and you can select only one. The default is "Frontal-Delaunay" (algorithm value 6), which balances quality and speed. The options are: "MeshAdapt" (1), "Automatic" (2), "Initial mesh only" (3), "Delaunay" (5), "Frontal-Delaunay" (6), "BAMG" (7), "Frontal-Delaunay for Quads" (8), "Packing of Parallelograms" (9), and "Quasi-structured Quad" (11). Each corresponds to a numeric value written to the 'Mesh.Algorithm' parameter in the generated '.geo' files. This setting affects how Gmsh meshes 2D surfaces before volume meshing, influencing triangle quality and distribution on topography surfaces.

3D Algorithm Selection

The "3D Algorithm" option selects the algorithm for 3D volume meshing. Six options are available, and you can select only one. The default is "Delaunay" (algorithm value 1), a standard method for tetrahedral meshing. The options are: "Delaunay" (1), "Initial mesh only" (3), "Frontal" (4), "MMG3D" (7), "R-tree" (9), and "HXT" (10). Each corresponds to a numeric value written to the 'Mesh.Algorithm3D' parameter in the generated '.geo' files. This setting controls how Gmsh generates tetrahedral elements in the 3D volume, affecting element quality, mesh density distribution, and generation speed. The choice depends on your geometry complexity and quality requirements.

Mesh Optimize Option

The "Mesh Optimize" option controls post-generation optimization to improve tetrahedral element quality. Four levels are available, and you can select only one. The default is "More Advanced" (value 2), which provides a good balance. The options are: "No Optimization" (0), "Basic Optimization" (1), "More Advanced" (2), and "The Most Aggressive" (3). Each corresponds to a numeric value written to the 'Mesh.Optimize' parameter in the generated '.geo' files. Optimization adjusts element shapes to improve quality metrics, reduce sliver elements, and enhance numerical stability. Higher levels take longer but generally improve quality.

Mesh Optimize Threshold

The “Mesh Optimize Threshold (0-1)” field sets the quality threshold below which tetrahedra are optimized. The default is 0.3. This value is written to the ‘Mesh.OptimizeThreshold’ parameter in the generated ‘.geo’ files. It should be between 0 and 1, where lower values optimize more elements (including better ones) and higher values optimize only the worst elements. The default of 0.3 focuses optimization on elements that need improvement while avoiding unnecessary processing of already good elements.

Mesh Smoothing

The “Mesh Smoothing” field sets the number of smoothing steps applied to the final mesh. The default is 10. This value is written to the ‘Mesh.Smoothing’ parameter in the generated ‘.geo’ files. Smoothing iteratively adjusts node positions to improve element quality and reduce distortion. More steps can improve quality but increase processing time. The default of 10 is typically sufficient for most applications, but you can increase it for challenging geometries or decrease it for faster processing.

How GMSH Settings Are Used

These settings are written into the generated ‘.geo’ files during STAGE 11 (mesh.geo generation) and used when GMSH runs. The software creates commands in the ‘.geo’ files that set ‘Mesh.Algorithm3D’ for 3D meshing, ‘Mesh.Algorithm’ for 2D meshing, ‘Mesh.Optimize’ for optimization level, ‘Mesh.OptimizeThreshold’ for the optimization threshold, and ‘Mesh.Smoothing’ for smoothing iterations. When GMSH processes these files, it uses these parameters to generate the mesh according to your specifications. The settings affect mesh quality, element distribution, generation time, and suitability for numerical simulations.

Integration with Mesh Generation Process

The GMSH settings are applied at multiple stages. In STAGE 3, GMSH meshes the initial topography surfaces (‘outlines_base.geo’ and ‘outlines_topo.geo’) using the specified 2D algorithm. In STAGE 11, GMSH generates the final 3D mesh from ‘mesh.geo’ using the 3D algorithm, optimization, and smoothing settings. It also converts the final 3D mesh from ‘*.msh’ to Tetgen files (i.e., .node, .ele, .neigh, .face, .edge, .t2e, .t2f, .f2e, and/or .vtu) if requested. The path you specify is used to locate and execute the Gmsh binary for all these operations. These settings are essential for controlling the mesh generation process and ensuring the output meets your requirements for geophysical modeling and inversion applications.

4. REGIONS

The REGIONS section in the Regions tab defines the spatial domains for the mesh. It includes four region types: COI (Core), ROI (Inner Padding), POI (Outer Padding), and Air. Each region can be defined as “None”, “Box” (rectilinear coordinates), or “From File” (irregular boundary file), except Air, which is controlled by a checkbox. These regions form the nested structure of the mesh, with COI at the center, surrounded by ROI, then POI, and optionally Air above the topography.

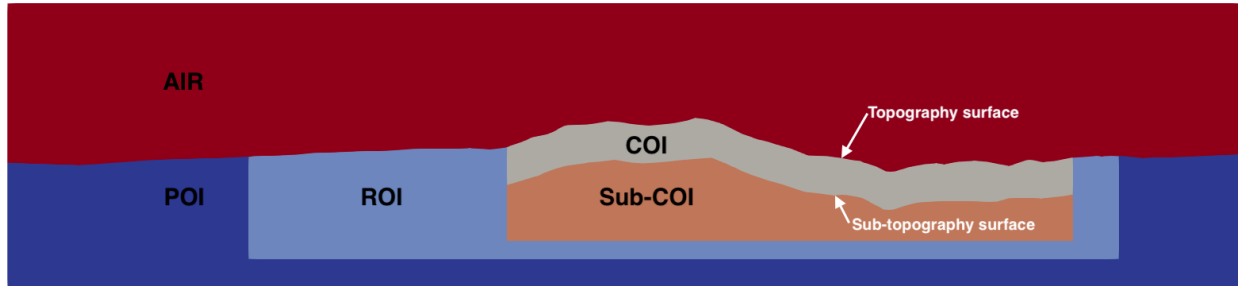


Figure 4: Regions/volumes for “Apply Standard” case.

Region Type Selection

For COI, ROI, and POI, three radio buttons control how the region is defined. “None” (default) excludes the region from the mesh. “Box” enables coordinate inputs to define a rectangular region using six values: x1, x2, y1, y2, z1, and z2. “From File” enables a file input to load an irregular boundary from a text file (the one generated in the REGION OUTLINE section). The selection determines which input fields are active; inactive fields are grayed out.

4.1. COI (Core) Region

COI is the innermost region, typically the area of interest. It can be defined as “None”, “Box”, or “From File”. When “Box” is selected, enter x1, x2, y1, y2, z1, and z2. When “From File” is selected, provide a text file with Zmin and Zmax on the first line, followed by X Y [Z] coordinate pairs defining the boundary. The file format should have Zmin and Zmax on the first line, then X Y [Z] pairs for each boundary point. You must also specify “Triangle Edge Size On Surface” (for topography surface meshing) and “Tetrahedron Edge Size In Volume” (for volume meshing). These control mesh cell size in the COI region.

4.2. Sub-COI Region

The COI section includes a “Sub-COI Region” subsection, available only when COI is defined (not “None”). It adds a sub-region below the topography within COI. The “Add Sub-COI” checkbox (unchecked by default) enables this feature. When unchecked, the Sub-COI fields are disabled and grayed out. When checked, three fields become active: “Depth to The Sub-Topography From Topography” (depth below topography for the sub-region base), “Triangle Edge Size On Surface” (for the sub-region surface mesh), and “Tetrahedron Edge Size In Volume” (for the sub-region volume mesh). It will add a surface similar to the topography

surface (following topography pattern but with different cell size; named 'sub-topography') to the COI region and split COI into two regions. The upper one is kept as COI and the lower one is considered as Sub-COI. Note: If “Fixed” is chosen for topography, Sub-COI is ignored, as variable topography is required for depth calculations.

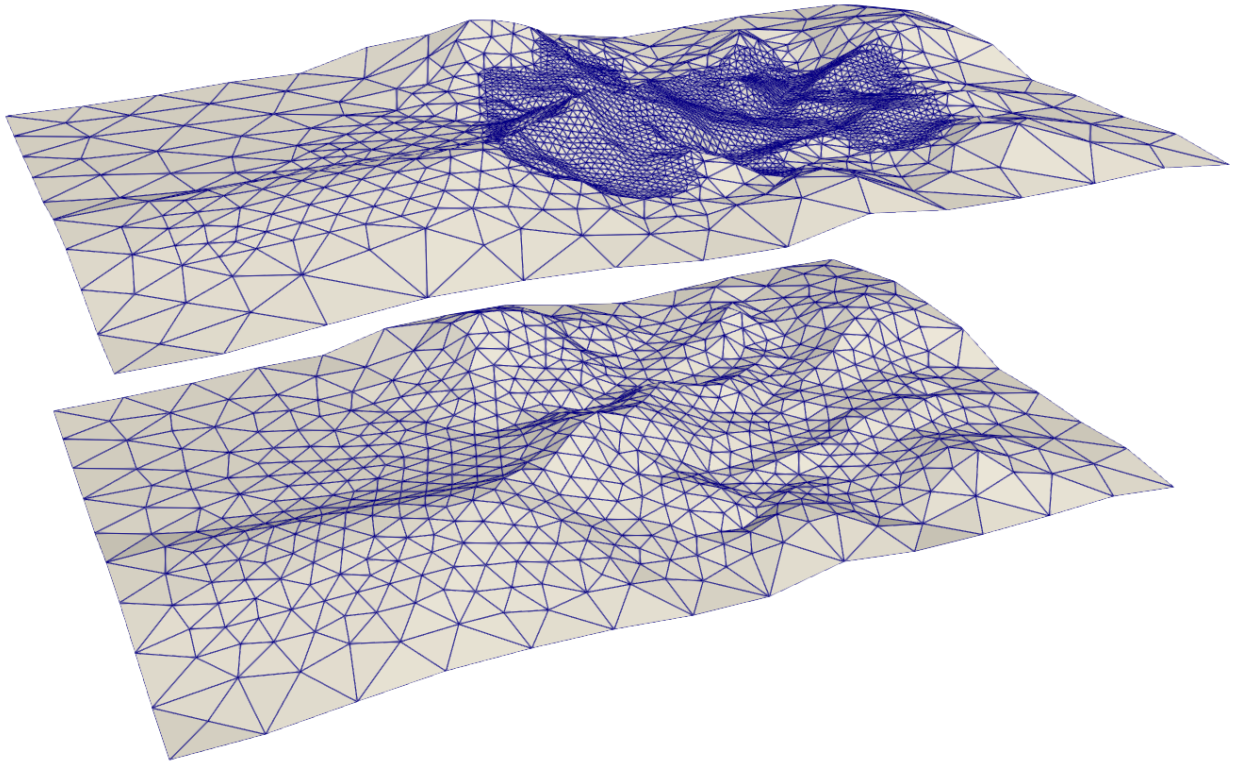


Figure 5: Topography (top) and sub-topography (bottom) surfaces. The refined region on the topography surface belongs to COI.

4.3. ROI (Inner Padding) Region

ROI is the region between COI and POI, providing padding around the core. It can be defined as “None”, “Box”, or “From File”, with the same coordinate and file input options as COI. When “Box” is selected, enter the six coordinate values. When “From File” is selected, provide a text file with the same format as COI (Zmin Zmax on first line, then X Y [Z] pairs). You must also specify “Triangle Edge Size On Surface” and “Tetrahedron Edge Size In Volume” for mesh sizing in this region.

4.4. POI (Outer Padding) Region

POI is the outermost region, providing additional padding around ROI. It can be defined as “None”, “Box”, or “From File”, with the same input options as the other regions. When “Box” is selected, enter the coordinate ranges. When “From File” is selected, provide a boundary file in the same format. You must also specify “Triangle Edge Size On Surface” and “Tetrahedron Edge Size In Volume” for mesh sizing.

4.5. Air Region

The Air Region is handled differently from the other regions. It does not use “None”, “Box”, or “From File” options. Instead, it is controlled by the “Model Air” checkbox. When unchecked (default), the Air region is not included. When checked, the Air region is included above the topography surface, and the “Tetrahedron Edge Size In Volume” field becomes active for specifying mesh cell size in the air volume. The Air region is generated above the outermost available region. For example, if only the COI and ROI regions are defined, the Air region will be placed above the ROI. The upper boundary of the Air region (which must lie above the topography) is determined by “z2” when the outer region is defined using Box, or by “Zmax” when From File is selected.

4.6. Refinement Around Topography

This includes four “In Air (Above Topography)”, “In COI (Below Topography)”, “In ROI (At COI Boundary)”, and “In POI (At ROI Boundary)” subsections with four refinement parameters for each.

Use “In Air” only when the “Model Air” is checked. For the “In Air” option, these parameters create a smooth transition in mesh size at the boundary between COI and Air, avoiding sharp transitions that can cause numerical issues. For “In Air”, we have “Minimum Cell Size” (recommended to match COI's Tetrahedron Edge Size In Volume), “Maximum Cell Size” (recommended to match Air's Tetrahedron Edge Size In Volume), “Refinement Radius” (recommended to start at about four times COI's Tetrahedron Edge Size In Volume), and “Transition Factor” (controls transition rate from min to max cell size; recommended to start at 2.0).

For the “In COI” option, these parameters also create a smooth transition in mesh size from the topography surface down to depth. This should be used when the “Triangle Edge Size On Surface” (in the COI Region section) is significantly smaller than the “Tetrahedron Edge Size In Volume”. Without this transition, the discrepancy can cause poorly shaped, high-aspect-ratio (long-cell) tetrahedra directly beneath the topography surface. Therefore, these parameters allow you to maintain smaller cells near the surface that gradually increase in size as they extend into the subsurface. For “In COI”, we also have “Minimum Cell Size” (recommended to match COI's Triangle Edge Size On Surface), “Maximum Cell Size” (recommended to match COI's Tetrahedron Edge Size In Volume), “Refinement Radius” (is depend on your need), and “Transition Factor” (controls transition rate from min to max cell size; recommended to start at 2.0).

When using the “In ROI (At COI Boundary)” and “In POI (At ROI Boundary)” options, the refinement parameters create a smooth transition in mesh size at the interface between the COI and ROI, or the ROI and POI, respectively. These settings are particularly important when the cell sizes between adjacent regions differ significantly, as a sharp discrepancy can result in poorly shaped, high-aspect-ratio (long-cell) tetrahedra at the boundary. For these two options, we also have “Minimum Cell Size” (recommended to match the Tetrahedron Edge Size In Volume of COI for “In ROI” and ROI for “In POI”), “Maximum Cell Size” (recommended to match the

Tetrahedron Edge Size In Volume of ROI for “In ROI” and POI for “In POI”), “Refinement Radius” (is depend on your need), and “Transition Factor” (controls transition rate from min to max cell size; recommended to start at 2.0).

Important Considerations

The regions form a nested structure: COI is innermost, surrounded by ROI, which is surrounded by POI. If you define ROI, COI should also be defined. If you define POI, both ROI and COI should typically be defined. The edge size parameters for each region control mesh density, with smaller values creating finer meshes and larger values creating coarser meshes. It is important to choose appropriate edge sizes to balance mesh quality with computational requirements. The Triangle Edge Size On Surface affects the surface mesh quality, while the Tetrahedron Edge Size In Volume affects the 3D volume mesh quality. These values should be chosen based on the scale of features you want to resolve in your geophysical modeling and inversion applications.

Also, NOTE that the Zmax of POI should be above ROI, and for the ROI it should be above COI, and for the COI it should be above topography.

5. SURFACE REFINEMENT

The Surface Refinement section in the Surface Refinement tab controls mesh refinement on the topography surface. It includes two subsections: “Refinement Around Stations” and “Refinement Along Lines/Loops”, plus a “Decimate” section. These options add refinement points on the surface to improve mesh resolution around stations or along lines, which is useful for geophysical surveys.

5.1. Refinement Around Stations

The “Refinement Around Stations” subsection refines the mesh around station locations on the topography surface (e.g., IP/DC resistivity and MT stations). It requires an “Input .node File” containing station coordinates in standard ‘.node’ format with a header line (number_of_nodes dimension 0 0) followed by node data (node_index x y z). For each station, three additional points forming a triangle are added around it at a spacing equal to the specified “Minimum Cell Size”.

Four parameters control the refinement behavior. “Minimum Cell Size” defines the distance between points surrounding each station and the smallest cell size in the refined region. “Maximum Cell Size” is the largest cell size at the edge of the refinement radius; it is recommended to be about twice the “Triangle Edge Size On Surface” of the COI region. “Refinement Radius” sets the distance from each station where refinement is applied. “Transition Factor” controls the transition rate from minimum to maximum cell size within the refinement radius; it should be greater than 0, with larger values producing sharper transitions.

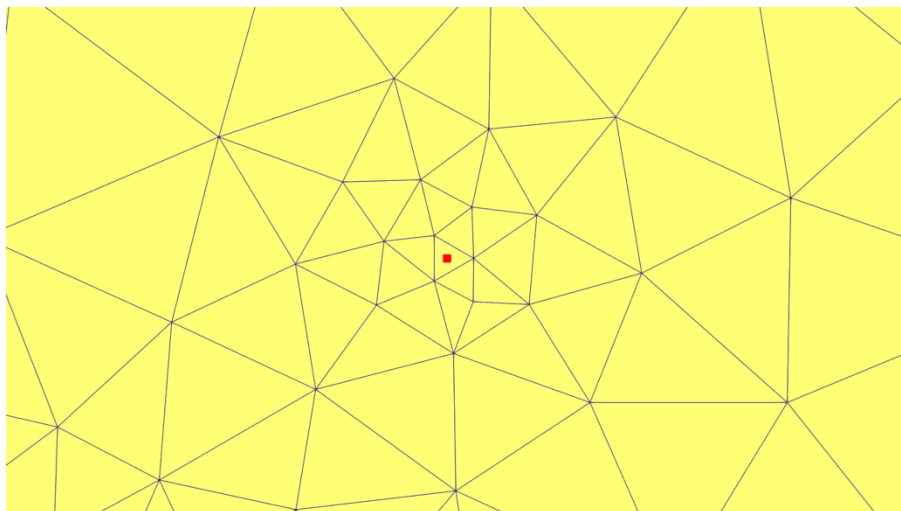


Figure 6: Refinement around a station (red dot) on the topography surface.

Extra Refinement Nodes Option for Stations

The “Specify Extra Refinement Nodes (In Volume)” option extends refinement into the volume above or below the surface around the stations. Three options are available: “None” (default, no volume refinement), “Above (In Air)” (refinement in the Air region above the topography), and “Below (In COI)” (refinement in the COI region below the topography). When “None” is selected, the extra refinement parameter fields are disabled and grayed out. When “Above” or “Below” is selected, four additional parameters become active (“Minimum Cell Size”, “Maximum Cell Size”, “Refinement Radius”, and “Transition Factor”), and a point/node will be added above/below the station at the same distance of “Minimum Cell Size”. The four parameters control the volume refinement behavior and are written to the generated ‘mesh.geo’ file in STAGE 11, creating distance fields that refine the mesh in the specified volume region around the point/node and consequently the stations.

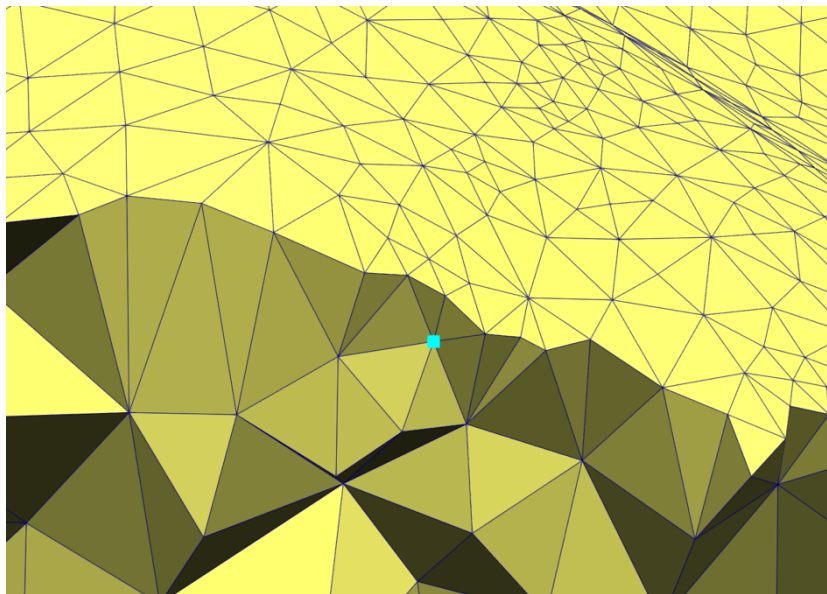


Figure 7: Extra refinement node (blue dot) under the topography surface where the station refinement triangle is located.

Interpolate On Topography Option for Stations

The “Interpolate On Topography” checkbox, when enabled, interpolates the station coordinates onto the topography surface during STAGE 12. This ensures stations align with the actual terrain elevation rather than remaining at their original Z coordinates. This option is useful when station files have fixed elevations that need to be adjusted to match the topography. If this option is selected, the “Output Type” in the Output Settings must be set to ‘all outputs’, and “Model Air” should not be selected, otherwise an error will be displayed when clicking the “Run” button. If you require both interpolated stations and the Air region, first run the process without the Air region selected. Once the interpolated stations have been generated, uncheck that option and re-run the process to create the final mesh including the Air region.

5.2. Refinement Along Lines/Loops

The “Refinement Along Lines/Loops” subsection refines the mesh along lines or loops on the topography surface (e.g., ground CSEM lines/loops). It requires an “Input.node File” containing nodes representing either the centers of loops or the vertices of lines/loops on the surface. These nodes are used to refine the mesh around the specified lines/loops by adding additional points at a spacing equal to the “Minimum Cell Size”.

The lines refinement uses the same four parameters as station refinement: “Minimum Cell Size” (distance between points surrounding lines/loops), “Maximum Cell Size” (recommended to be about twice the “Triangle Edge Size On Surface” of the COI region), “Refinement Radius” (distance from lines/loops where refinement is applied), and “Transition Factor” (controls transition rate from min to max cell size, should be greater than 0, with larger values producing sharper transitions).

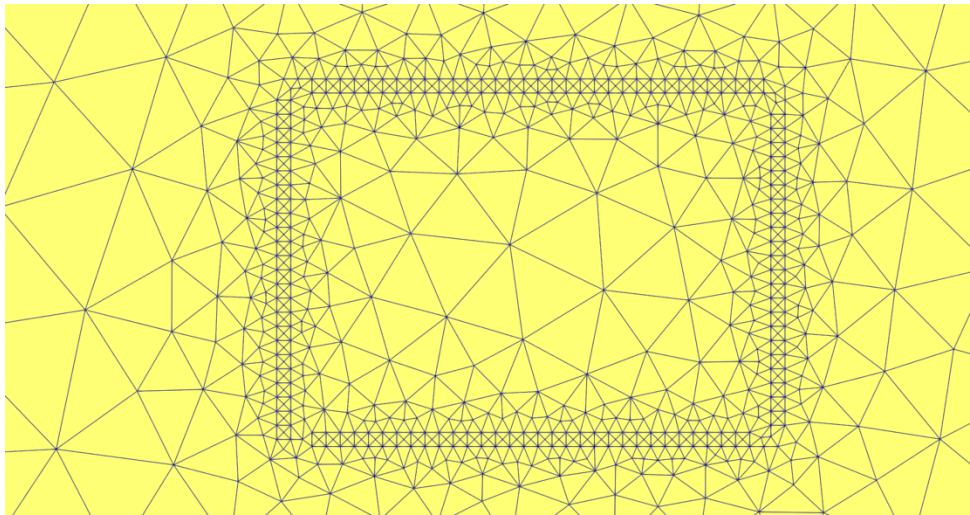


Figure 8: Refinement along a line/loop on the topography surface.

File Type

The “File Type” option determines how the input file is interpreted. “Centers” (default) means each node represents the center of a loop, and the software generates square loops around these center points based on the specified “Loop Size”. When “Centers” is selected, the “Loop Size (Square)” field becomes active, allowing you to specify the edge length of the square loop around each node. The “Line Type” field is disabled and grayed out when “Centers” is selected. “Vertices” means the nodes represent the vertices of lines or loops. When “Vertices” is selected, the “Loop Size” field is disabled and grayed out, and the “Line Type” field becomes active.

When “Vertices” is selected for File Type, the “Line Type” option becomes active with three choices. “Continuous (Segment)” (default) treats the vertices as points along continuous line segments from start to end point. “Continuous (Loop)” treats the vertices as points along continuous loops, where the last point connects back to the first point to form a closed loop.

“Discrete (Segment)” treats pairs of consecutive nodes as separate line segments, where each pair forms one independent line segment (start point and end point).

Input File Formats

Centers Mode:

```

number_of_nodes dimension 0 0
node_index1 x1 y1 z1
node_index2 x2 y2 z2
...
node_indexn xn yn zn

```

- Each node represents the center of a loop
- The software generates loops around these center points based on the specified Loop Size

Vertices Mode (Discrete Segment):

```

number_of_nodes dimension 0 0
node_index1 x1 y1 z1   Start point of line 1
node_index2 x2 y2 z2   End point of line 1
node_index3 x3 y3 z3   Start point of line 2
node_index4 x4 y4 z4   End point of line 2
node_index5 x5 y5 z5   Start point of line 3
node_index6 x6 y6 z6   End point of line 3
...

```

- 2 vertices per line (start and end points)
- Each pair of consecutive nodes forms one line segment

Vertices Mode (Continuous Segment):

```

number_of_nodes dimension 0 1
node_index1 x1 y1 z1 1 First point of line1
node_index2 x2 y2 z2 1 Second point of line1
node_index3 x3 y3 z3 1 Third point of line1
node_index4 x4 y4 z4 2 First point of line2
node_index1 x5 y5 z5 2 Second point of line2
node_index2 x6 y6 z6 2 Third point of line2
node_index3 x7 y7 z7 2 Fourth point of line2
...
node_indexn xn yn zn m nth point of linem

```

- vertices along line (start point to end point)
- Fifth column indicate each line's nodes; if not available, all nodes belong to one line

Vertices Mode (Continuous Loop):

```

number_of_nodes dimension 0 1
node_index1 x1 y1 z1 1 First point of loop1
node_index2 x2 y2 z2 1 Second point of loop1
node_index3 x3 y3 z3 1 Third point of loop1

```

```

node_index4 x4 y4 z4 2 First point of loop2
node_index1 x5 y5 z5 2 Second point of loop2
node_index2 x6 y6 z6 2 Third point of loop2
node_index3 x7 y7 z7 2 Fourth point of loop2
...
node_indexn xn yn zn m nth point of loopm

```

- it connects `node_indexn` to `node_index1` (end point to start point) to make a loop
- Fifth column indicate each loop's nodes (>2); if not available, all nodes belong to one loop

Extra Refinement Nodes Option for Lines

The “Specify Extra Refinement Nodes (In Volume)” option for lines works the same way as for stations. You can choose “None” (default), “Above (In Air)”, or “Below (In COI)”. When “Above” or “Below” is selected, four additional parameters become active (“Minimum Cell Size”, “Maximum Cell Size”, “Refinement Radius”, and “Transition Factor”), and two series of points/nodes will be added above/below the line/loop at the same distance of “Minimum Cell Size”. The four parameters control the volume refinement behavior and are written to the generated ‘mesh.geo’ file in STAGE 11, creating distance fields that refine the mesh in the specified volume region around the points/nodes and consequently along the lines/loops.

Interpolate On Topography Option for Lines

The “Interpolate On Topography” checkbox for lines works the same way as for stations. When enabled, it interpolates the point coordinates in the node file onto the topography surface during STAGE 12, ensuring lines follow the actual terrain elevation. The same requirements apply: “Output Type” must be set to '0', and “Model Air” should not be selected.

Decimate Section

The “Decimate” section includes a “Decimate Refinement Nodes On Surface (Distance)” field. This removes refinement nodes that are too close to each other based on a maximum separation distance. For any two nodes separated by less than the specified distance, the second node is removed. This reduces the number of refinement points and helps avoid very close nodes that can increase the number of cells unnecessarily. The recommendation is to set this value to be less than or equal to the smallest “Minimum Cell Size” used in the surface refinement sections. If not provided, the software uses the smaller of the “Minimum Cell Size” values from the station and lines refinement sections.

6. VOLUME REFINEMENT

The Volume Refinement section in the Volume Refinement tab controls mesh refinement within the 3D volume. It includes three subsections: “Refinement In Volume” (stations and lines), “Refinement Of Zones”, and “Decimate”. These options add refinement points inside the volume to improve mesh resolution around stations, along lines/loops, or within specific zones.

Refinement In Volume Section

The “Refinement In Volume” section includes two subsections: “Refinement Around Stations” and “Refinement Along Lines/Loops”. These work similarly to their surface counterparts but operate inside the volume rather than on the topography surface. Each subsection has two refinement sets labeled “1st” and “2nd”, allowing you to define refinement in more regions or with different parameters.

6.1. Refinement Around Stations In Volume

The “Refinement Around Stations” subsection refines the mesh around station locations inside the volume (e.g., airborne FDEM and airborne MT). To provide more options for refinement, it includes two subsections: “1st” and “2nd”. Each requires an “Input .node File” containing station coordinates in standard ‘.node’ format with a header line (number_of_nodes dimension 0 0) followed by node data (node_index x y z). Each subsection (“1st” and “2nd”) includes a “Regions” selection with three radio buttons: “COI”, “ROI”, and “Air”. This selection specifies the volume/region in which the stations are located. For each station, four main additional points forming a tetrahedron are added around it at a spacing equal to the “Minimum Cell Size” specified below. Eight extra points will be also located around the tetrahedron to help with better refinement if “Add Extra Refinement Points Around Stations” is checked. The refinement transitions from “Minimum Cell Size” to “Maximum Cell Size” within the “Refinement Radius” using the “Transition Factor” specified below.

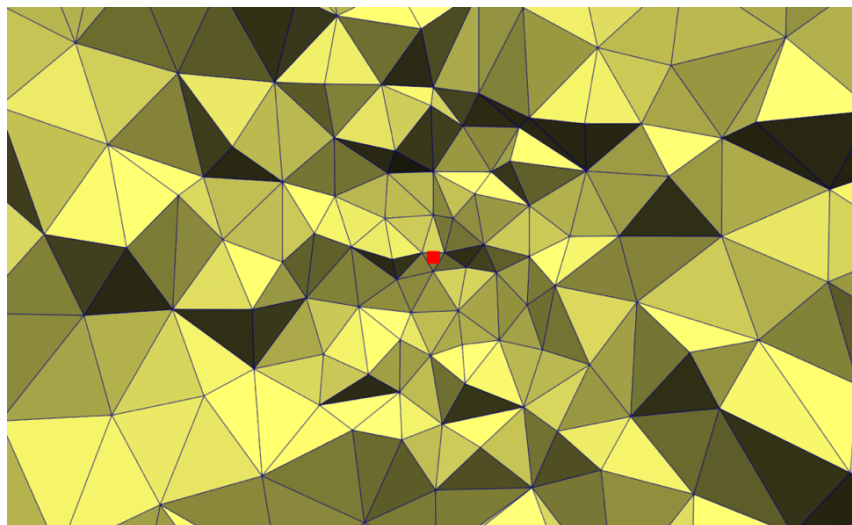


Figure 9: Refinement around a station (red dot) in the volume.

Four parameters control the refinement behavior for each subsection. “Minimum Cell Size” defines the distance between points surrounding each station and the smallest cell size in the refined region. “Maximum Cell Size” is the largest cell size at the edge of the refinement radius; it is recommended to be about twice the “Tetrahedron Edge Size In Volume” of the selected region. “Refinement Radius” sets the distance from each station where refinement is applied. “Transition Factor” controls the transition rate from minimum to maximum cell size within the refinement radius; it should be greater than 0, with larger values producing sharper transitions.

6.2. Refinement Along Lines/Loops In Volume

The “Refinement Along Lines/Loops” subsection refines the mesh along lines or loops inside the volume (e.g., airborne TDEM). To provide more options for refinement, it includes two subsections: “1st” and “2nd”. Each requires an “Input .node File” containing nodes representing either the centers of loops or the vertices of lines/loops. These nodes are used to refine the mesh around the specified lines/loops by adding additional points (around the lines/loops) at a spacing equal to the “Minimum Cell Size”. The refinement transitions from “Minimum Cell Size” to “Maximum Cell Size” within the “Refinement Radius” using the “Transition Factor”. Each subsection (“1st” and “2nd”) includes a “Regions” selection with three radio buttons: “COI”, “ROI”, and “Air”. This selection specifies the volume/region in which the lines/loops are located.

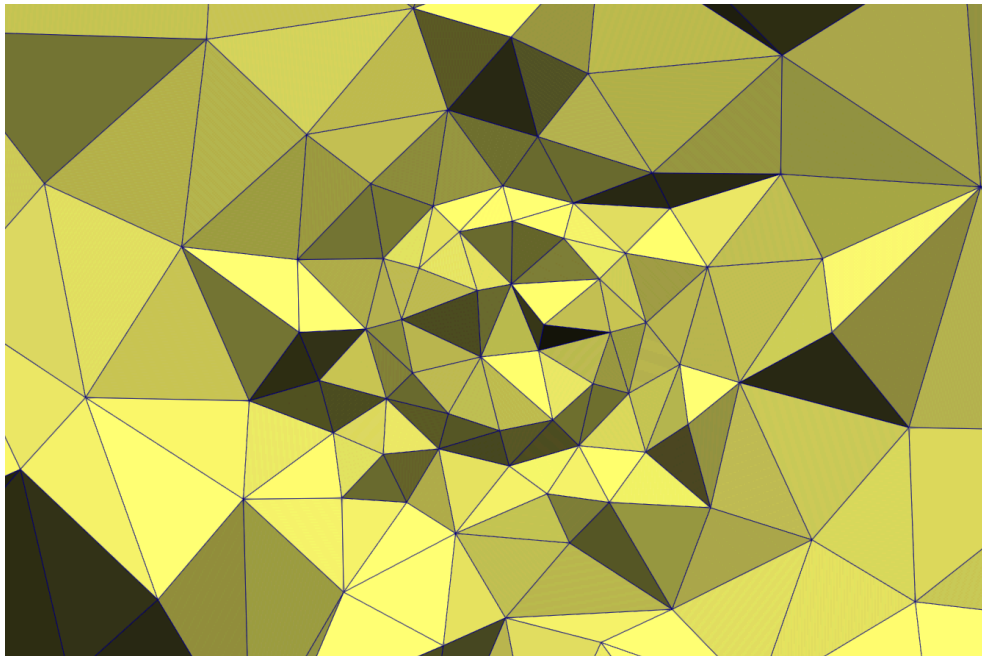


Figure 10: Refinement along a loop in the volume.

File Type

The “File Type” option determines how the input file is interpreted for volume lines refinement. “Centers” (default) means each node represents the center of a loop, and the software generates loops around these center points based on the specified “Loop Type” and “Loop Size”. When

“Centers” is selected, the “Loop Type” and “Loop Size” fields become active, and the “Line Type” field is disabled and grayed out. “Vertices” means the nodes represent the vertices of lines or loops. When “Vertices” is selected, the “Loop Type” and “Loop Size” fields are disabled and grayed out, and the “Line Type” field becomes active.

When “Centers” is selected for File Type, the “Loop Type” option becomes active with three choices. “Square” (default) generates square loops around each center point. “Octagon” generates octagonal (8-sided) loops around each center point. “Dodecagon” generates dodecagonal (12-sided) loops around each center point. The “Loop Size” field specifies the edge length of the loop, which applies to all three loop types.

When “Vertices” is selected for File Type, the “Line Type” option becomes active with three choices, similar to surface lines refinement. “Continuous (Segment)” (default) treats the vertices as points along continuous line segments from start to end point. “Continuous (Loop)” treats the vertices as points along continuous loops, where the last point connects back to the first point to form a closed loop. “Discrete (Segment)” treats pairs of consecutive nodes as separate line segments, where each pair forms one independent line segment (start point and end point).

Input File Formats

Centers Mode:

```

number_of_nodes dimension 0 0
node_index1 x1 y1 z1
node_index2 x2 y2 z2
...
node_indexn xn yn zn

```

- Each node represents the center of a loop
- The software generates loops around these center points based on the specified Loop Size

Vertices Mode (Discrete Segment):

```

number_of_nodes dimension 0 0
node_index1 x1 y1 z1   Start point of line 1
node_index2 x2 y2 z2   End point of line 1
node_index3 x3 y3 z3   Start point of line 2
node_index4 x4 y4 z4   End point of line 2
node_index5 x5 y5 z5   Start point of line 3
node_index6 x6 y6 z6   End point of line 3
...

```

- 2 vertices per line (start and end points)
- Each pair of consecutive nodes forms one line segment

Vertices Mode (Continuous Segment):

```

number_of_nodes dimension 0 1
node_index1 x1 y1 z1 1 First point of line1
node_index2 x2 y2 z2 1 Second point of line1

```

```

node_index3 x3 y3 z3 1 Third point of line1
node_index4 x4 y4 z4 2 First point of line2
node_index1 x5 y5 z5 2 Second point of line2
node_index2 x6 y6 z6 2 Third point of line2
node_index3 x7 y7 z7 2 Fourth point of line2
...
node_indexn xn yn zn m nth point of linem

```

- vertices along line (start point to end point)
- Fifth column indicate each line's nodes; if not available, all nodes belong to one line

Vertices Mode (Continuous Loop):

```

number_of_nodes dimension 0 1
node_index1 x1 y1 z1 1 First point of loop1
node_index2 x2 y2 z2 1 Second point of loop1
node_index3 x3 y3 z3 1 Third point of loop1
node_index4 x4 y4 z4 2 First point of loop2
node_index1 x5 y5 z5 2 Second point of loop2
node_index2 x6 y6 z6 2 Third point of loop2
node_index3 x7 y7 z7 2 Fourth point of loop2
...
node_indexn xn yn zn m nth point of loopm

```

- it connects node_indexn to node_index1 (end point to start point) to make a loop
- Fifth column indicate each loop's nodes (>2); if not available, all nodes belong to one loop

6.3. Refinement Of Zones Section

The “Refinement Of Zones” section refines the mesh around points within a specific volume region. Note that the points themselves will not be included in mesh. It requires an “Input .node File” containing points that define zones where refinement should be applied. The file should be in standard ‘.node’ format with a header line (number_of_nodes dimension 0 0) followed by node data (node_index x y z). The “Regions” selection includes three radio buttons: “COI”, “ROI”, and “Air”. This selection specifies the volume/region in which the points are located. All points in the input file must be located within the selected region; if points are outside the selected region, they will be ignored or may cause errors.

Four parameters control the refinement behavior. “Minimum Cell Size” defines the smallest cell size in the refined region around each point. “Maximum Cell Size” is the largest cell size at the edge of the refinement radius; it is recommended to be set to the “Tetrahedron Edge Size In Volume” of the selected region. “Refinement Radius” sets the distance from each point where refinement is applied; it is recommended to be greater than or equal to “Minimum Cell Size”, and possibly a number between “Minimum Cell Size” and “Maximum Cell Size”. “Transition Factor” controls the transition rate from minimum to maximum cell size within the refinement radius; it should be greater than 0, with larger values producing sharper transitions. It is recommended to start with a value of 2.0 and adjust until you obtain the desired mesh.

6.4. Decimate Section

The “Decimate” section includes a “Decimate Refinement Nodes In Volume (Distance)” field. This removes refinement nodes that are too close to each other based on a maximum separation distance. For any two nodes separated by less than the specified distance, the second node is removed. This reduces the number of refinement points and helps avoid very close nodes that can increase the number of cells unnecessarily. The recommendation is to set this value to be less than or equal to the smallest “Minimum Cell Size” used in the volume refinement sections. If not provided, the software uses the smaller of the “Minimum Cell Size” values from the volume station and lines refinement sections.

7. MESH OPERATORS

The Mesh Operators section in the Mesh Operators tab controls how surfaces are combined during mesh generation. It includes two methods: "Apply Standard" (default) and "Apply Boolean", plus an "Import Extra Geometry Files" subsection for importing additional geometries when using Boolean mode.

7.1. Apply Standard Method

"Apply Standard" (default) uses a standard method for mesh generation. Surfaces intersect where lines and nodes already exist. This method is faster than Boolean for large meshes and is suitable for most cases. When selected, the "Import Extra Geometry Files" subsection is disabled and grayed out, as extra geometries are only used with Boolean operations.

7.2. Apply Boolean Method

"Apply Boolean" uses Boolean operators to combine surfaces. The operator intersects surfaces and automatically generates nodes and lines at intersections. This is slower than Standard for large meshes, but useful when you need precise intersections or want to include extra geometries.

When "Apply Boolean" is selected, the workflow differs from "Apply Standard" starting at STAGE 11. Before STAGE 11, the process is the same. In STAGE 11, '.geo' files for COI, ROI, and POI regions (if available) are made. The software then generates five '.geo' files that are processed in sequence using the Boolean Assistant dialog to finally generate the 'mesh.msh' file. Note that "Apply Boolean" intersects COI, ROI, and POI (if any are available) with the topography surface. Thus, it always generates an Air region which can be removed by the user if not needed.

Boolean Assistant:

After generating the five '.geo' files, the user opens the Boolean Assistant from the "Output Settings" tab. The Boolean Assistant is a step-by-step dialog that guides the user through running Gmsh on each '.geo' file, configuring parameters, and generating the final mesh — all from within TETRIUM's GUI. The five steps are:

Step 1: Generate initial .brep (File: `1_PreMesh_Booleab_All.geo`)

This file intersects all volumes and surfaces using BooleanFragments and saves the result as `1_premesh_boolean_all.brep`. Depending on the number of points, this may take minutes or even hours (e.g., 50,000+ points). The Boolean operator creates new volumes with different IDs from the originals.

The file includes all necessary input '.geo' files:

- Region files: '.geo' files for COI, ROI, and POI regions (if available) made for "Apply Boolean".

- Topography surface: The topography surface file ('outlines_topo.geo' or 'outlines_topo_int.geo' if available).
- Base surface: The base surface file (e.g., 'outlines_base_int_s.geo' if generated).
- Imported surface files: Any imported surface geometry files (not points) specified in the "Import Extra Geometry Files" section of the Mesh Operators tab.

Configurable parameters (via the Boolean Assistant):

- Geometry.Tolerance: Maximum spatial distance allowed when identifying identical geometrical entities such as points, edges, or surfaces. It determines when entities are close enough to be considered identical, essential for removing duplicate points.
- Geometry.ToleranceBoolean: Same concept applied specifically to the Boolean operation.

Output: `1_premesh_boolean_all.brep`

Step 2: Preview mesh to identify volume IDs (File: `2_PreMesh_All.geo`)

Creates a quick 3D mesh (`2_premesh_all.vtk`) so the user can open it in ParaView and identify the volume IDs (i.e., CellEntityIds in ParaView) of each region (COI, Air, etc.). If no volumes need to be removed, Steps 3 and 4 should be skipped and the user should go directly to Step 5.

Output: `2_premesh_all.vtk`

Step 3: Remove unwanted volumes and re-generate .brep (File: `3_PreMesh_Boolean.geo`)

Similar to Step 1 but includes Recursive Delete commands to remove volumes the user does not need (e.g., Air). Volume IDs to remove are identified from the VTK output of Step 2.

Configurable parameters (via the Boolean Assistant):

- Geometry.Tolerance and Geometry.ToleranceBoolean
- Volume IDs to remove (comma-separated list)

Output: `3_premesh_boolean.brep`

Step 4: Verify updated volume IDs (File: `4_PreMesh_All.geo`)

Creates a quick mesh (`4_premesh_all.vtk`) from the pruned .brep so the user can verify that the volume IDs have changed and identify the new IDs for Step 5.

Output: `4_premesh_all.vtk`

Step 5: Final mesh generation (File: `5_PreMesh.geo`)

Generates the final `mesh.msh` and `5_premesh.vtk` with full mesh refinement. This step has the most configurable parameters:

- Considered Step: Select "1" if Steps 3 and 4 were skipped (uses the .brep from Step 1), or "2" if they were completed (uses the .brep from Step 3).
- Geometry.Tolerance
- Volume Identities (table): Lists all volume IDs present in the mesh and assigns to each an attribute ID (>100; can be changed later to physical property values) and a cell edge size for the tetrahedrons in that volume.
- Refinements (table): Assigns the correct volume ID to each refinement point type (e.g., "Refinement Around Stations (Surface)", "Refinement At Boundary", etc.) based on the volume in which the refinement points are located. Leave blank if a refinement type is not available.
- Imported Points (table): If point files were imported in the "Mesh Operators" tab, assigns the correct volume ID to each imported point set.
- Mesh algorithms: 3D algorithm (Delaunay, Frontal, HXT, etc.) and 2D algorithm (MeshAdapt, Delaunay, Frontal-Delaunay, etc.).
- Mesh optimization: Smoothing steps, optimize level, and optimize threshold.

Output: `mesh.msh` and `mesh.vtk`

Boolean Assistant — Dialog Controls

Each step tab in the Boolean Assistant provides the following controls:

- Show .geo File: Opens the corresponding '.geo' file in the system text editor for viewing or manual editing.
- Save Changes: Applies the parameter values from the GUI widgets into the '.geo' file on disk. Always click this before running Gmsh if parameters have been changed.
- Verbosity: Selects the level of information Gmsh prints to the log (0 = silent, 1 = errors, 2 = warnings, 3 = direct, 4 = information, 5 = status (default), 99 = debug).
- Run Gmsh on Step N: Executes Gmsh on the step's '.geo' file. Gmsh runs in a background thread so the GUI remains responsive, and output is displayed in real time in the log panel.
- Stop: Terminates a running Gmsh process.
- Log panel: Displays real-time Gmsh output for the current step.
- Save Settings: Saves all Boolean Assistant parameters into a '.ttrm' file (merged with existing settings if the file already exists). Settings are automatically loaded from the most recent '.ttrm' file in the output directory when the dialog is opened.

Important Considerations

- Copy imported files: Before running Step 1, copy all imported point and surface '.geo' files (especially those brought in via the "Mesh Operators" tab) to the output directory where the other '.geo' files are located.

- GMSH SDK path: The Gmsh executable path is taken from the "GMSH Path" setting in the TETRIUM main window. The Boolean Assistant resolves the path to `/bin/gmsh` (or `gmsh.exe` on Windows) automatically.
- Cross-platform: The Boolean Assistant works on macOS, Windows, and Linux. On Windows, Gmsh runs without spawning a console window.

After the Boolean Assistant

After generating `mesh.msh` via Step 5 of the Boolean Assistant, use the "Post-Run" option in the Output Settings tab to continue. When "Run" is clicked with both "Apply Boolean" and "Post-Run" selected, the software skips RUN 11.0, 11.1, and 11.2, and proceeds directly to RUN 11.3 (conversion from mesh.msh to Tetgen format files: `.node`, `.ele`, etc.).

7.3. Import Extra Geometry Files

The "Import Extra Geometry Files (*.geo)" subsection becomes active only when "Apply Boolean" is selected. When "Apply Standard" is selected, this subsection is disabled and grayed out. This subsection allows you to import up to 99 additional geometry files (`.geo` format) that will be included in the Boolean operations. These files can contain points or surfaces/volumes that you want to incorporate into your mesh. For each of the 99 file inputs (labeled "1st Input File" through "99th Input File"), you can specify two geometry types using radio buttons:

Points: (default) The imported file contains point geometries. These points will be included in the Boolean operations and can be used to refine the mesh or add specific features.

Surfaces/Volumes: The imported file contains surface or volume (closed surfaces) geometries. These surfaces or volumes will be included in the Boolean operations and will be intersected with the main mesh geometry.

Important Considerations for Imported Geometries

When importing extra geometry files, it is critical that the index/offset of points and surfaces in the imported geometries are different from any other indices in the mesh. If there are conflicts, the Boolean operations may fail or produce incorrect results. You should ensure that your imported geometries use different offset values to avoid conflicts and overlaps each other (e.g., e.g., larger than 20,000,000; increments of 1,000,000).

You can generate geological surfaces (such as body or contact surfaces) directly within ParaView using the 'Geometric Shapes' tool. Once created, save these models as .OBJ files. You can then use the 'OBJ to Geo' program to convert them into the required .Geo format. You can also import your own geological models from other software. We support standard 'triangulated' surface formats such as .TS, .DXF and .OBJ. These can be converted into the required .Geo format directly (e.g., using program "OBJ to Geo") or via a step-by-step method: 1. Use the 'Convert Format' program to generate node/ele files, 2. Use the 'Mesh to Geo' program to convert those node/ele files into the final .geo format.

The Gmsh Boolean operator does not support parallel processing. Consequently, surfaces with high node counts (often found in models imported from external software such as Leapfrog, GOCAD, and so on) will result in excessively long runtimes. These imported surfaces (in formats such as .ts, .obj, .stl, .dxf) can be frequently over-refined or contain poorly shaped cells that hinder performance. To optimize the workflow, you must refine your surfaces by reducing the number of nodes and cells while retaining essential geological detail. For this, convert your imported surface files to node/ele format using the 'Convert Format' program in ZAMINEX. To be able to refine in ParaView, first transform node/ele files into .vtu format using 'Mesh to VTU'. For MeshLab, convert them to .obj using 'Convert Format'. Finally, use the decimation or simplification tools in ParaView or MeshLab to reduce complexity. For MeshLab, see 'Appendix A' in `paraview_tutorial.pdf`.

When to Use Each Method

Use "Apply Standard" when you need fast mesh generation for large models and when the standard surface intersection method is sufficient. This is the recommended method for most applications. Use "Apply Boolean" when you need precise control over geometry intersections, when you want to include extra geometries in your mesh (e.g., in forward modelling or constrained inversions), or when the standard method does not produce the desired results. The Boolean method is particularly useful for complex geological models where precise intersection of multiple surfaces is critical. Note that it is slower than the standard method.

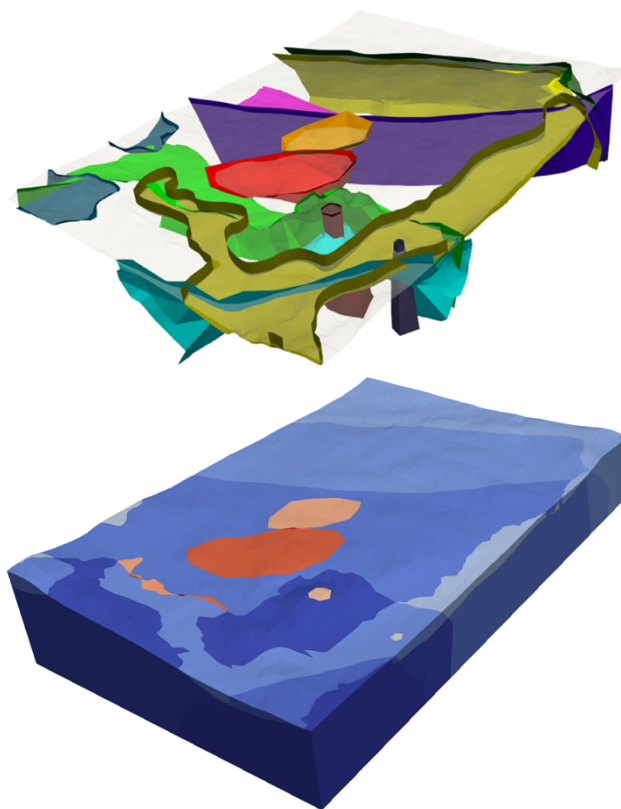


Figure 11: Imported geometries (top) and their mesh (bottom) generated using Boolean.

8. OUTPUT SETTINGS

The OUTPUT SETTINGS section in the Output Settings tab controls output file generation, naming, location, and execution options. It includes Output Type, Boundary Marker Value, Root Name For Output Files, Output Location, Keep Intermediate Output Files, and Post-Run.

8.1. Output Type

Output Type selects which files are generated. Five radio button options:

- “File .msh”: Stops after Gmsh execution in STAGE 11. Generates only ‘mesh.msh’ and does not convert to Tetgen format. Use this to inspect the Gmsh mesh before conversion.
- “Files .msh, .vtu, .node, .ele”: Generates ‘.msh’, ‘.vtu’, ‘.node’, and ‘.ele’. The ‘.vtu’ is for visualization, ‘.node’ contains node coordinates, and ‘.ele’ contains tetrahedron connectivity.
- “Files .msh, .vtu, .node, .ele, .neigh”: Adds ‘.neigh’ (neighbor information for each tetrahedron).
- “Files .msh, .vtu, .node, .ele, .neigh, .face, .edge”: Adds ‘.face’ (boundary faces) and ‘.edge’ (boundary edges).
- “Files .msh, .vtu, .node, .ele, .neigh, .face, .edge, .t2e, .t2f, .f2e”: Generates all files, including ‘.t2e’ (tetrahedron-to-edge), ‘.t2f’ (tetrahedron-to-face), and ‘.f2e’ (face-to-edge).

Important Note: If the last option (including all files) is selected, the topography files (i.e., .node, .ele, and .vtu) generated from the final mesh will be created. Please note that the model must not include an Air region.

Important Note: If “Interpolate On Topography” is selected in Surface Refinement, “Output Type” must be set to “all files”, and “Model Air” must not be selected. Otherwise, an error is shown when clicking “Run”.

8.2. Boundary Marker Value

Boundary Marker Value is an integer used for boundary markers in the ‘.face’ file. Default is “1”. This value labels boundary faces in the mesh and is used by geophysical modeling/inversion software to identify boundary conditions. You can change it to any integer if your software requires a different marker value.

8.3. Root Name For Output Files

Root Name For Output Files is the prefix for all output files. If you enter “mesh”, the software creates ‘mesh.node’, ‘mesh.ele’, ‘mesh.vtu’, etc. If left empty, it defaults to “mesh”. This name is used by ‘gmsh2tetgen.py’ in STAGE 11 to name the converted files. Choose a name that reflects your project or model.

8.4. Output Location

Output Location is the directory where all output files are saved. Use the “Browse” button to select a folder. If not specified, files are saved in a temporary directory. All generated files (including intermediate files if “Keep Intermediate Output Files” is selected) are written to this location. Ensure you have write permissions and sufficient disk space. You must always consider a separate and independent folder for the outputs.

8.5. Keep Intermediate Output Files

Keep Intermediate Output Files is a checkbox (unchecked by default). When checked, all intermediate files from the mesh generation process are preserved. When unchecked, intermediate files are deleted at the end, except for:

- ‘outlines_base.geo’: to see the mesh refinement on the flat sub-topography surface
- ‘outlines_base.vtu’: to see the mesh refinement on the flat sub-topography surface
- ‘outlines_topo.geo’: : to see the mesh refinement on the flat topography surface
- ‘outlines_topo.vtu’: to see the mesh refinement on the flat topography surface
- ‘outlines_base_int.vtu’: to see the mesh refinement on the sub-topography surface
- ‘outlines_base_int.ele’: to see the mesh refinement on the sub-topography surface
- ‘outlines_base_int.node’: to see the mesh refinement on the sub-topography surface
- ‘outlines_topo_int.vtu’: to see the mesh refinement on the topography surface
- ‘outlines_topo_int.ele’: to see the mesh refinement on the topography surface
- ‘outlines_topo_int.node’: to see the mesh refinement on the topography surface
- All files from STAGES 11 and 12 (i.e., mesh files as well as interpolated node files)

After STAGE 3, you can stop the run and examine the mesh refinement quality on the topography surface. If the results are not satisfactory, adjust the parameters and rerun the process. Note that at this stage the topography surface is still represented as flat.

Important: If you plan to use “Post-Run”, you must check this option in the first run to preserve the intermediate files needed for Post-Run.

8.6. Post-Run

Post-Run is a checkbox (unchecked by default). When selected, execution starts from STAGE 11 instead of STAGE 1. This is useful after a full run when you want to modify certain parameters and re-run only the final stages.

How Post-Run Works with Apply Standard

When “Apply Standard” is selected, Post-Run starts from STAGE 11 (mesh.geo generation). This assumes you have completed a full run (at least until running GMSH in STAGE 11) and have all necessary intermediate files. You can then modify parameters and re-run only STAGES 11 and 12 for 3D mesh generation.

Important Requirements for Post-Run (Apply Standard):

- For the first run, “Keep Intermediate Output Files” must be selected to preserve the intermediate files needed for Post-Run.
- The ONLY parameters that can be modified for Post-Run are:
 - All “Tetrahedron Edge Size In Volume” values (for COI, ROI, POI, Air, Sub-COI)
 - “Minimum Cell Size” values (for all refinement sections)
 - “Maximum Cell Size” values (for all refinement sections)
 - “Refinement Radius” values (for all refinement sections)
 - “Transition Factor” values (for all refinement sections)
- All other parameters (topography, regions, refinement files, etc.) cannot be changed in Post-Run mode, as the intermediate files from the previous run are used.

How Post-Run Works with Apply Boolean

When “Apply Boolean” is selected, Post-Run starts from STAGE 11, RUN 11.3 (conversion from mesh.msh to Tetgen files). This assumes you have:

- Completed a full run that generated the 5 boolean .geo files
(‘1_PreMesh_Booleab_All.geo’, ‘2_PreMesh_All.geo’, ‘3_PreMesh_Boolean.geo’, ‘4_PreMesh_All.geo’, ‘5_PreMesh.geo’)
- Run GMSH manually on these .geo files to generate ‘mesh.msh’
- The ‘mesh.msh’ file exists in the output directory

When you click “Run” with both “Apply Boolean” and “Post-Run” selected, the code skips RUN 11.0, 11.1, and 11.2, and proceeds directly to RUN 11.3 (conversion from mesh.msh to Tetgen files). This allows you to manually run GMSH on the boolean .geo files and then use the software to convert the resulting ‘mesh.msh’ to Tetgen format.

9. IMPORTANT NOTES

1- Before a new run after applying some changes, you might need to close and reopen “TETRIUM” and even delete all files in the output folder if you encounter errors or unwanted model. For example, when you receive errors like below in Gmsh runs:

“Error: Invalid boundary mesh (overlapping facets) on surface...”

2- You can check out the initial refinement on topography from file “outline_topo.vtk” (and “outline_base.vtk” if available for the base of COI), immediately after STAGE 3. It does not have variations but sufficient for refinement assessment. Note that these are ‘vtk’ files not ‘vtu’ files.

3- If “Fixed” is chosen for “Topography”, Sub-COI region will be ignored.

4- When using the 'In COI' for refinement cases, the recommended cell size used for refinement (e.g., Maximum Cell Size) should match the 'Tetrahedron Edge Size' defined for the COI. Otherwise, the resulting mesh in the COI region may deviate from your intended cell size.

5- If applying refinement, make sure you always consider a value for “Decimate” available at the bottom of refinement tabs.

5- If “Keep Intermediate Output Files” is not selected, all generated files will be deleted at the end except the following files IF AVAILABLE:

outlines_base.geo, outlines_base.vtu, outlines_topo.geo, outlines_topo.vtu, outlines_topo_int.vtu, outlines_topo_int.ele, outlines_topo_int.node, outlines_base_int.vtu, outlines_base_int.ele, outlines_base_int.node, and all files generated in STAGES 11 and 12.

6- If you see the following errors on macOS, you should give access to the Gmsh library in "Privacy & Security" of "System Settings". Scroll down to see a message stating "gmsh was blocked...". Click "Open Anyway".

*“Mesh generation failed
 Errors encountered:
 Error: Running GMSH on outlines_base.geo
 GMSH outlines_base.geo failed!”*

or

*“gmsh” cannot be opened because the developer cannot be verified.
 macOS cannot verify that this app is free from malware.*

or

*macOS cannot verify the developer of "gms". Are you sure you want to open it?
By opening this app, you will be overriding system security which can expose your
computer and personal information to malware that may harm your Mac or compromise
your privacy.*

7- You must always consider a separate and independent folder for your outputs.

8- The Zmax of POI should be above ROI, and for the ROI it should be above COI, and for the COI it should be above topography.

9- Using Mesh.Algorithm3D = 10 (HXT) is effective for maintaining mesh quality in regions with large size transitions, which can otherwise result in poorly shaped, high-aspect-ratio (long-cell) tetrahedra. However, ensure that the mesh quality remains appropriate in other areas of the model as well.

10- On Windows, gms might not be found due to a small bug in some of their SDK package. For this, download these two options from gms website:

(1)-Download Gms for Windows

(2)-Download the Software Development Kit (SDK) for Windows

Copy "gms.exe" from (1) and put it in the folder 'bin' of (2). Now, the gms SDK folder (i.e., (2)) is ready to use.

11- If you also require Voronoi mesh files, you will need to use external software such as Tetgen. First, use TETRIUM to generate the five necessary output files (e.g., mesh.node, mesh.ele, mesh.neigh, mesh.face, and mesh.edge). Then, use these as inputs for Tetgen and run the following command:

`./tetgen -rvefnk mesh` (or alternatively, `./tetgen -rvefnk mesh`)

10. EXAMPLES

10.1. Example for “Apply Standard” case

The attached “test_files” folder contains the file “standard.ttrm”, which stores all saved parameters and settings for this example. The remaining files provide the necessary input data for multiple geophysical scenarios, all applied to a single mesh for learning purposes.

Begin by loading “standard.ttrm” using the “Load Settings” menu. Note that the imported input files may need to be reloaded, as their saved file paths may not match your system. In this example, the topography data has already been imported and decimated to reduce the number of points, improving mesh generation speed. The Gmsh directory is set, and default parameters are used.

All regions (COI, ROI, POI, Air, and Sub-COI) are included. The irregular outlines for the COI and ROI are generated in the Region Outline section, while the POI region is defined using the Box option. The Air region is included, and a sub-topography surface is placed 400 m below the actual topography, dividing the COI into two volumes: the upper COI and the lower Sub-COI.

Refinement Options:

- Refinement around surface stations: suitable for IP/DC resistivity, MT, or surface (EM) base stations.
- Refinement along surface loops: suitable for ground TDEM loops.
- Refinement around stations in volume: used for airborne MT, FDEM, TDEM receivers, and boreholes.
- Refinement along loops in volume: used for airborne TDEM transmitters.

The refinement size depends on your project needs and is controlled through the Minimum Cell Size parameter. This manual and the software’s info buttons provide guidance on refinement settings. Keep in mind that additional refinement improves accuracy but increases runtime and memory usage. You can manage refinement levels using parameters such as Refinement Radius and Transition Rate. A recommended workflow is to run the mesh generation with initial values, then use Post-Run to adjust refinement settings without restarting the entire process.

Mesh visualization can be done using ParaView. Physical attributes assigned to each region begin at value 101, and can be viewed as “cell_attribute” in ParaView.

To evaluate refinement on the topography surface (especially when stations or loops are present) you may stop the process after STAGE 3 and inspect the files “outline_topo.vtk” (and “outline_base.vtk” if the Sub-COI is defined). At this stage the topography is still flat, but the files are sufficient for assessing refinement quality.

10.2. Example for “Apply Boolean” case

The same parameters are used for the “Apply Boolean” example. This example loads two additional geometry files: “point1.geo”, which contains points to be added to the model, and “surface1.geo”, which defines an additional surface.

During the first run, the process stops at STAGE 11 after generating five “.geo” files. These files must be manually executed by the user using “Boolean Assistant” to produce the “mesh.msh” file. Before starting the second run, make sure “mesh.msh” is in the output directory, and then select Post-Run.

For convenience, a saved setting file (i.e., boolean.ttrm) is provided in the “ttrm” folder to help you better understand this workflow.

In Boolean operations, the Air region is always present. Intersecting surfaces may create additional volumes, so it is necessary to identify these new volumes and adjust or remove them as needed within the five “.geo” files.