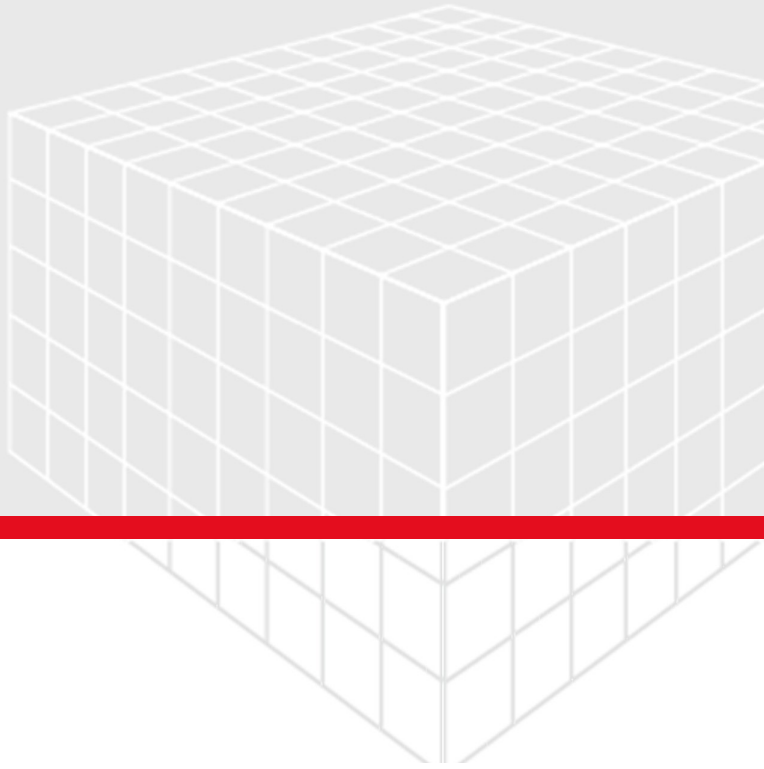
ExportGeo-CAD

User Guide



GEODICT Release 2026

Published: June 2026

GeoDict

https://doi.org/10.30423/userguide.geodict

ExportGeo-CAD

1 ExportGeo-CAD

Use ExportGeo-CAD to convert GeoDict data formats to a smooth surface mesh with sub-voxel precision and save it to your hard disk for further use with external software.

Meshes can be generated directly from

- structure files (*.gdt and *.gad files) which often result from the GeoDict strutcure generators,
- gray value images from imported µCT images,
- index images (*.g32) from structure analysis like Identify Grains,
- and volume fields (e.g., *.vap, *.hht, ...) which are results of property prediction simulations.

ExportGeo-CAD saves the created mesh directly to the hard disk. Therefore, it is not possible to edit the mesh after creation.

Know how! Use the MeshGeo module under **Model → MeshGeo** if you want to keep the mesh in memory after creation and do subsequent smoothing and coarsening operations. Afterwards, the mesh can be saved in different file formats directly from MeshGeo.

MeshGeo is included in your ExportGeo-CAD license.

New in GeoDict 2026

- New Export Object CAD 25 command to export the analytic objects or their inverse pore space geometry directly in a CAD format.
- Native CAD export (STEP/IGES) now works without additional third-party software.

In the **ExportGeo-CAD** section, to the left of the Visualization area, a pull-down menu displays the choices for the meshing options.

Learn to use ExportGeo-CAD:

- Export Voxel Mesh 5
  Create surface meshes from voxel data.
- Export Object Mesh 17
  Create triangulations from the analytic object data.
- Export Object CAD 25
  Create CAD formats from the analytic object data.

Join the conversation

Visit the Community on GeoDict Forum to be inspired and get answers to top questions.

Find illustrative application examples on the website.
Our tutorials and seminar videos give you a first hands-on experience. They are collected in the Learning Center.
Send your Support Request to our technical Support.

1.1 Export Voxel Mesh

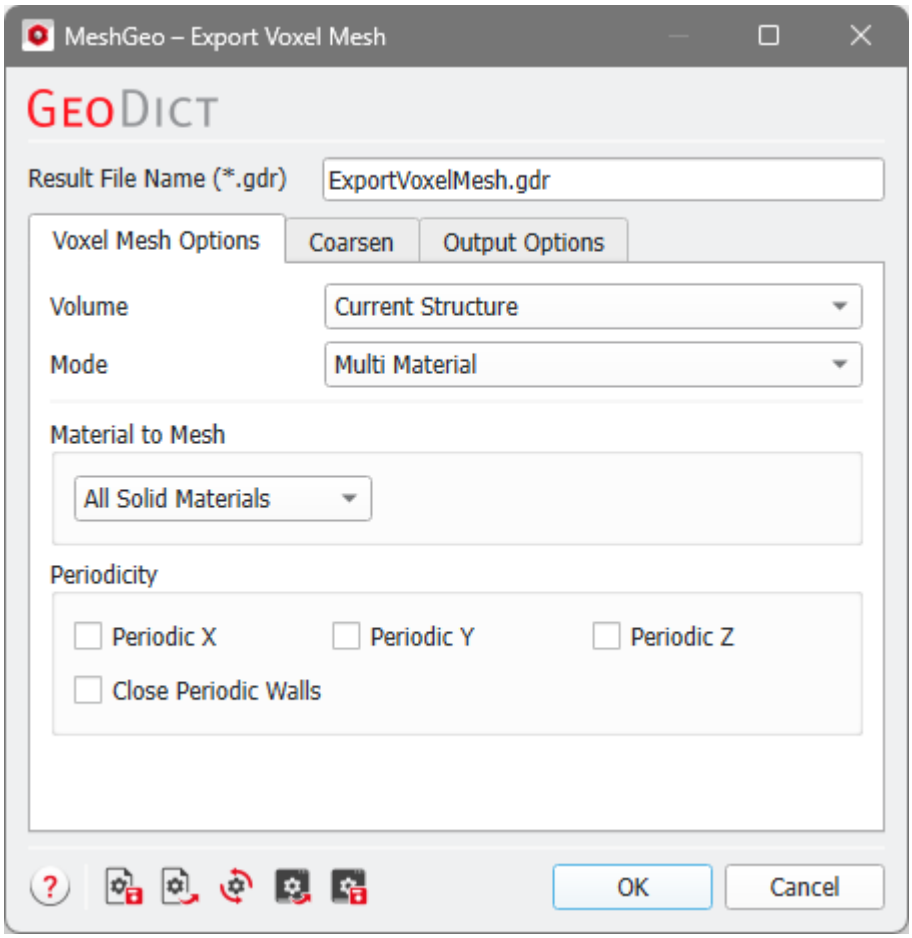
Meshes of a structure in GeoDict can be created based on the voxel information using **Export Voxel Mesh**. This voxel information can be a structure, a gray value image, an index image, or a volume field.

The Export Voxel Mesh options:

- Voxel Mesh Options 5
  Set the options for the creation of the mesh.
- Coarsen 11
  Decide if the mesh should be coarsened and select the options for coarsening.
- Output Options 14
  Determine the output file format.

1.1.1 Voxel Mesh Options

The **Voxel Mesh** options allow to create triangulations from the available voxel information. Additionally, the **Mode** defines how the selected volume is meshed. The choices for **Volume** and **Mode** change the other available options.



Volume

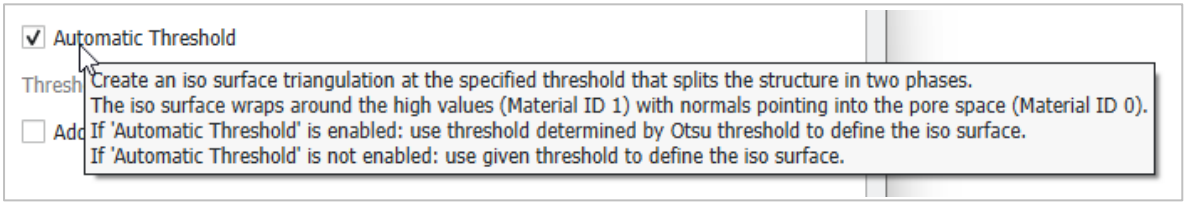
Different types of **Volume** structure information can be meshed: **Current Structure**, **Current Grey-Value Image**, **Index Image**, and **Volume Field**.

Current Structure

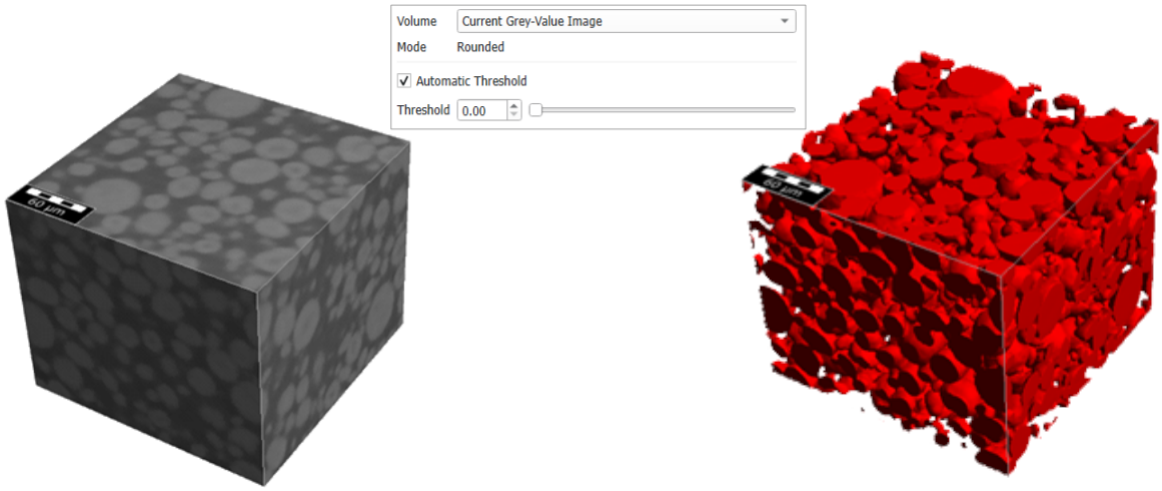
With this option, the currently loaded voxel structure is converted to a mesh. For Current Structure, all four **Mode**s are available: **Voxel Surface**, **Rounded**, **Smooth**, and **Multi Material**.

Current Grey-Value Image

Instead of first converting a gray-value image to a voxel structure (with ImportGeo-Vol), it is possible to directly mesh the **Current Grey-Value Image**. In this case, the **Mode** is fixed to **Rounded**. Based on the given **Threshold**, an isosurface is created that splits the structure in two phases. The normal of the isosurface points into the pore space. If **Automatic Threshold** is selected, the Otsu algorithm is used to determine the threshold. The tooltip also displays the explanation when the mouse hovers over **Automatic Threshold**.



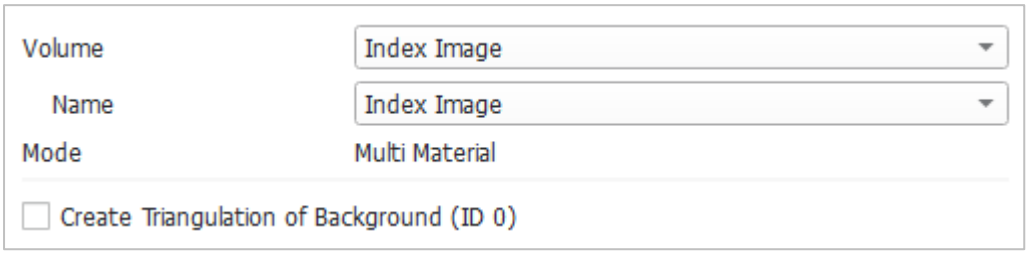
With this option, all information from the gray-value image can be used. The created mesh is smoother than a mesh created from a segmented voxel structure, since the gray-value information allows to obtain sub-voxel precision.



Index Image

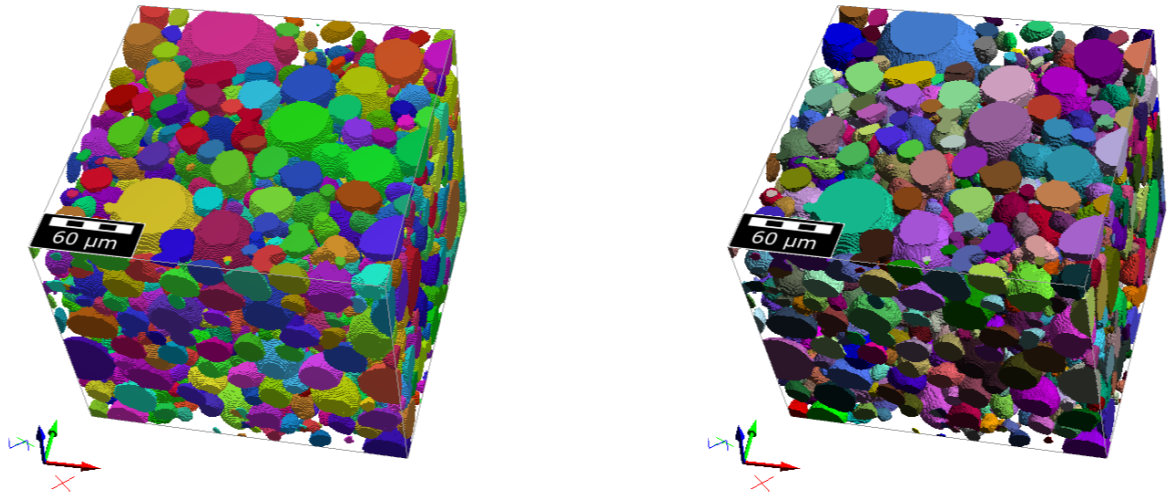
By choosing an **Index Image** as Volume, it is possible to use the object indices to create **Multi Material** triangulations. Analogously, the object indices from GrainFind or FiberFind results can be used in the same way.

Each object with a given index is converted to its own mesh, and the meshes for different objects do not overlap. The **Mode** is fixed to **Multi Material**, since this is the option that keeps the object information and preserves the partition of unity.



If **Create Triangulation of Background (ID 0)** is checked, the triangulation is created also for the Index 0 in the image, usually pore material.

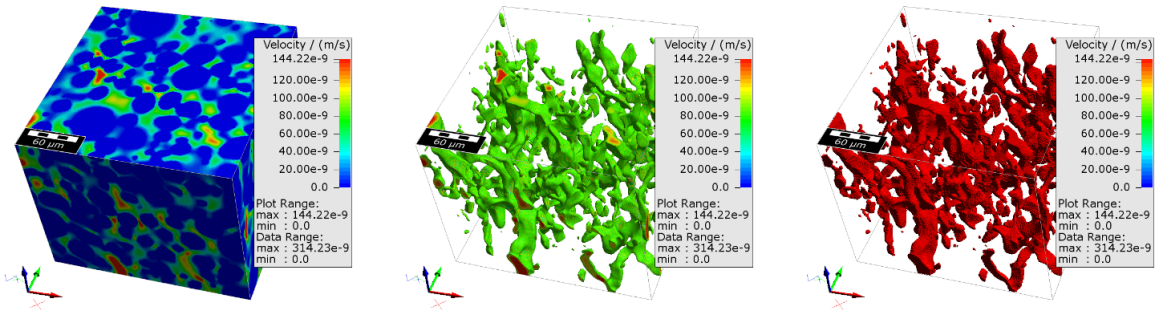
In the figure below, to the left an index image from a GrainFind result is shown. The right figure shows the mesh calculated from that index image.



Volume Field

With the **Volume Field** option, triangulations can be created from arbitrary GeoDict results. In the example below, a flow field from a FlowDict simulation is converted to a mesh. The **Volume Field** option works analogously to the **Current Grey-Value Image** option: It creates an isosurface mesh based on the given **Threshold**.

In the figure below, observe from left to right: A flow field calculated with FlowDict, the flow field with a threshold (i.e., only fast velocities are shown), and a mesh computed with the same threshold.



Mode

Four **Mode**s can be used to export voxel data: **Voxel Surface**, **Rounded**, **Smooth**, or **Multi Material**. **Multi Material** is the default. Which options are available depends on the selected option for **Volume**: For example, if the **Current Grey-Value Image** is selected, only **Rounded** is available.

Voxel Surface

Voxel Surface exports the exact surface of the voxel structure with its characteristic rectangular edges. This option is available in GeoDict to ensure compatibility with older releases. Use the **Voxel Surface** mode only if you intend to preserve the staircase-like pattern of voxel structure, otherwise, **Multi Material** mode is recommended.

Rounded

Rounded works analogously to the Smooth export, but it focuses on triangle quality instead of smoothness. The exported surface is constrained to the domain and the

surfaces of multiple materials may intersect. This export mode is most suitable for numerical simulations with only one material phase (e.g., flow simulations in the pore space).

Smooth

The **Smooth** algorithm is an improved version of the marching cubes algorithm which produces a mesh with a better triangle quality. The mesh is smoothed during creation. This surface is not constrained to the domain and the surfaces of multiple materials may intersect. **Smooth** is the most suitable export mode for visualization purposes.

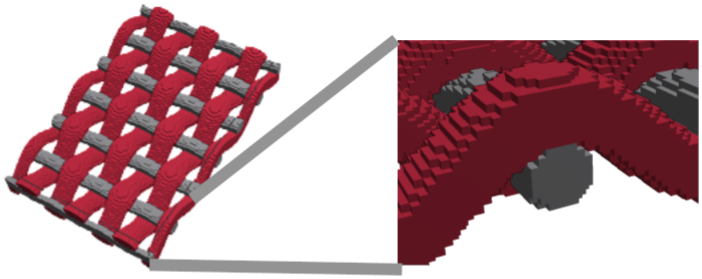
Multi Material

With the **Multi Material** option, the meshes for the different materials do not overlap. They have perfect contact at the points where different materials touch. When all materials are exported, a partition of unity is created. This means that each point in the domain is enclosed in exactly one surface. This property is preserved with further processing steps (e.g., **Smooth Mesh** or **Coarsen Mesh**) in MeshGeo.

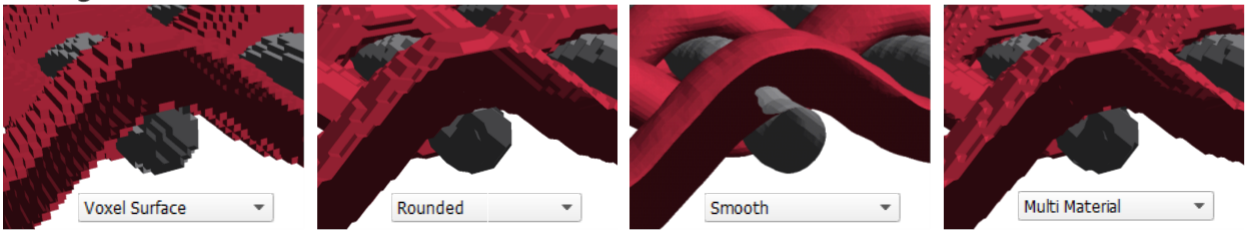
Multi Material meshes are constrained to the domain. This export mode is suitable to mesh multi-material microstructures for numerical simulations.

The differences of the four modes can be observed from this example:

Voxel Structure in GeoDict

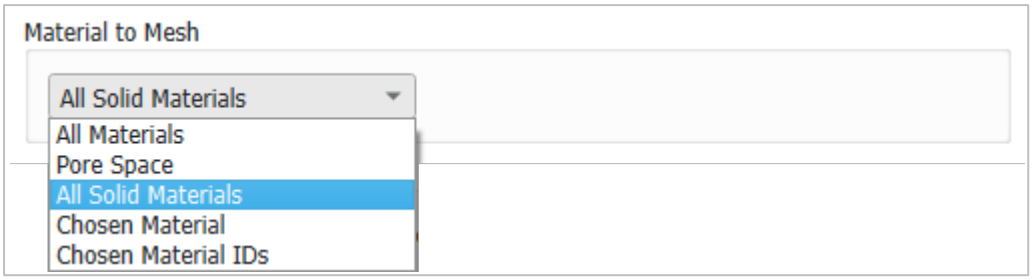


Triangulations in different modes

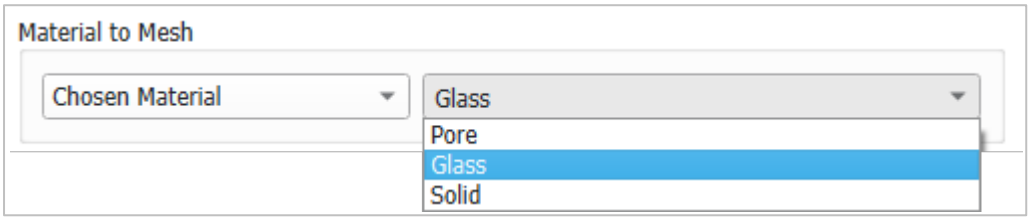


Material to Mesh

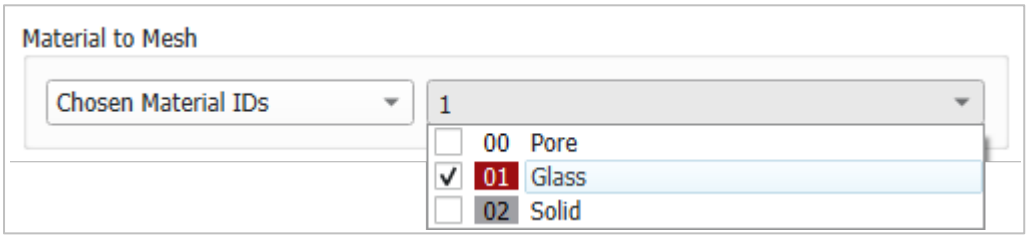
Select the Material IDs to be meshed with the options under **Material to Mesh**. With the first three options, either **All Materials**, the **Pore Space**, or **All Solid Materials** are meshed. **All Solid Materials** is the default, since this is the best choice for most use cases.



With **Chosen Material**, all Material IDs of the chosen material are meshed.

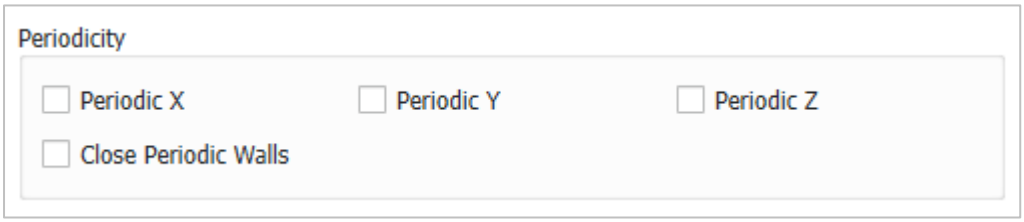


With **Chosen Material IDs**, individual Material IDs can be selected, which can have assigned different materials.



Periodicity

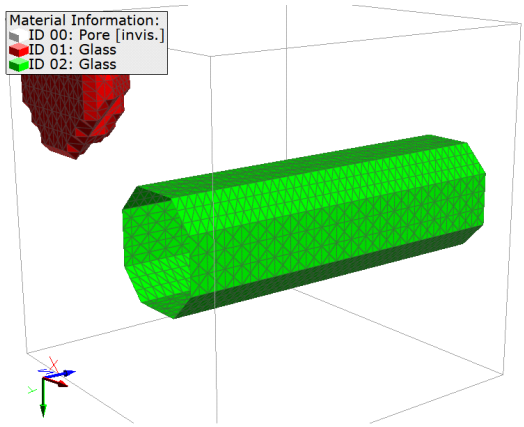
The boundary condition defines how the triangulation algorithm deals with boundary voxels.



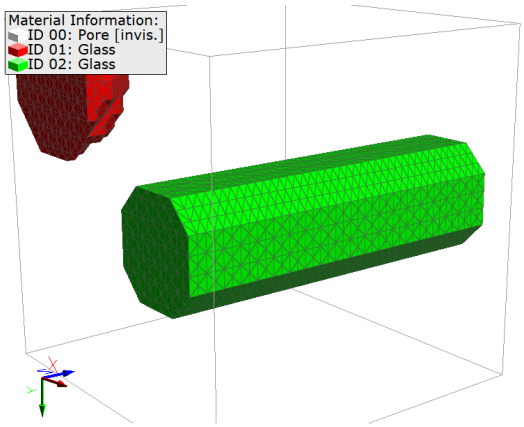
When **Periodicity** is selected, the mesh of one end is periodic to the other end of the structure in the chosen direction. This option should only be used if the original structure is periodic.

When **Periodicity** is not selected, symmetric boundary conditions are applied and the mesh is closed at all boundaries.

In the pictures below, the difference can be observed at the boundary in Z-direction. On the left, periodic boundary conditions are used, and the green mesh is not closed since it continues at the opposite side. On the right, symmetric boundary conditions are used.



Periodic

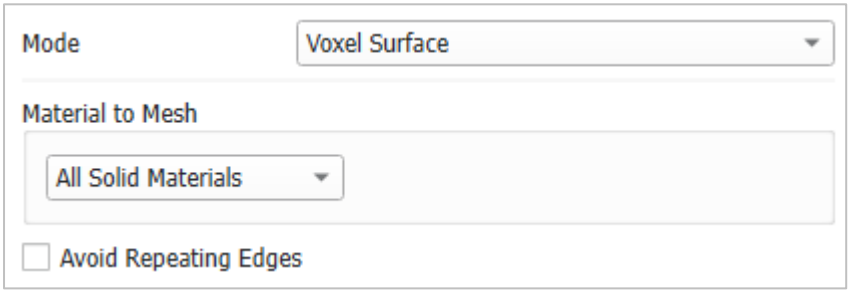


not Periodic

When **Periodicity** is selected for a direction, the mesh walls are by default open at the boundaries. With **Close Periodic Walls**, these periodic walls are closed.

The **Periodicity** option is only available for the **Multi Material** mode.

Avoid Repeating Edges

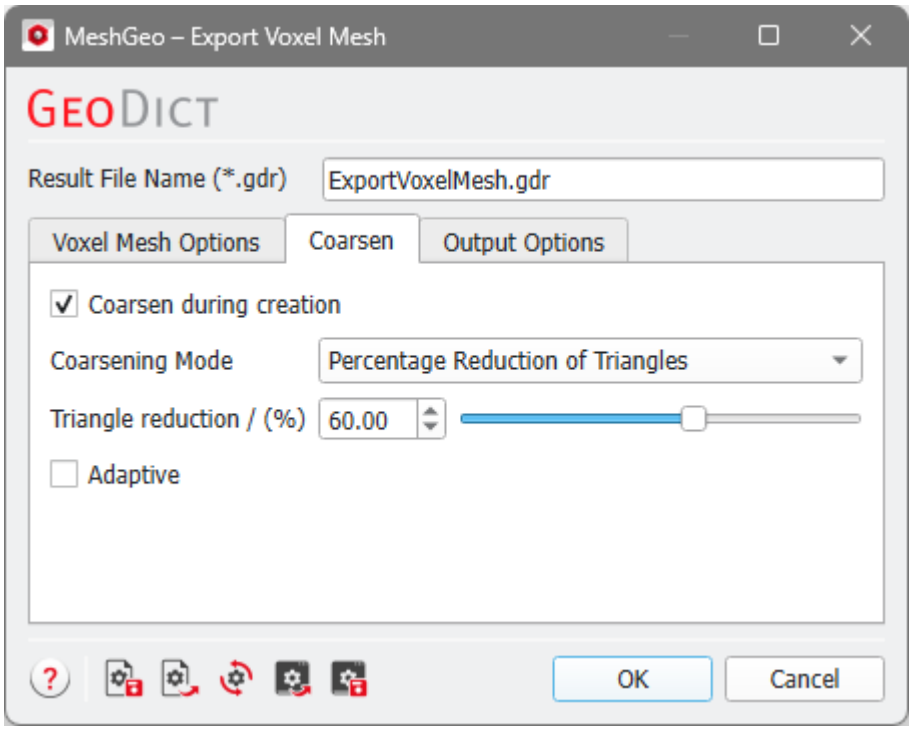


This option is available when **Voxel Surface** mode is chosen. Where different materials are in contact, their voxel surfaces share the same edges. This is not allowed in a mesh in the STL format. With **Avoid Repeating Edges**, these edges are shifted slightly inwards the material. Then, each material has different edges.

For the **Multi Material** option, such an option is not necessary since there, the different materials are saved to different meshes.

1.1.2 Coarsen

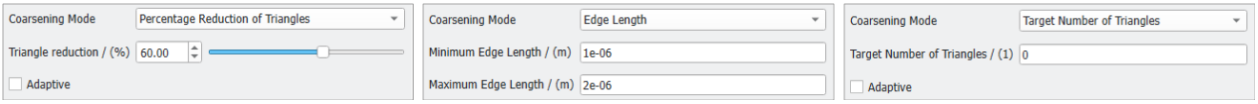
Check **Coarsen during creation** to enable coarsening during mesh creation.



Coarsening Mode

MeshGeo provides three modes to coarsen triangulations, based on the stopping criteria for the coarsening algorithm: by **Percentage Reduction of Triangles**, by **Edge Length**, and by **Target Number of Triangles**.

The coarsening algorithm reduces the number of triangles in the currently loaded mesh by combining short edges to longer edges. Depending on the choice of **Coarsening Mode**, the different stopping criteria and their corresponding input parameters are shown.

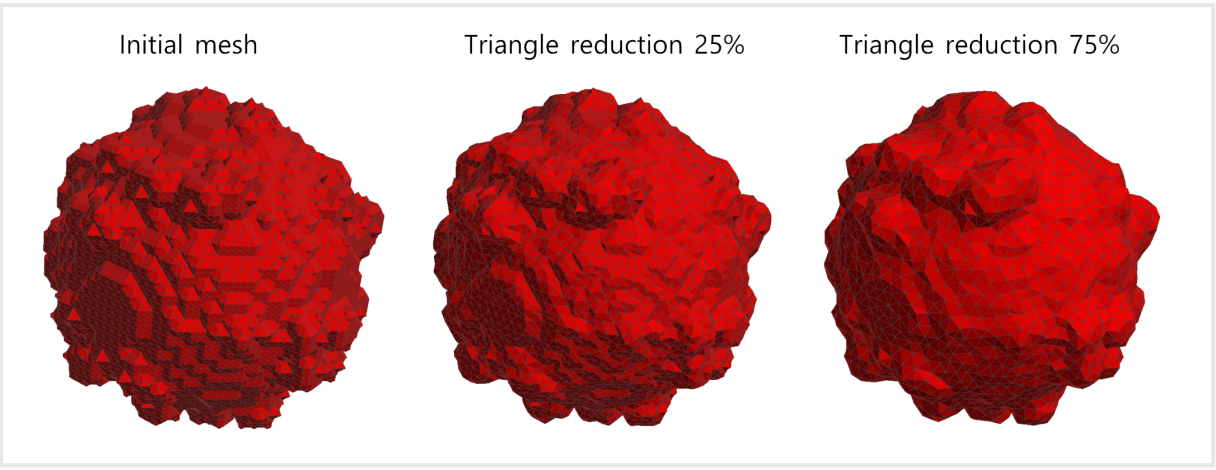


Percentage Reduction of Triangles

Percentage Reduction of Triangles removes at least the given percentage of triangles. The reduction of the triangle number is done with an iterative algorithm. The algorithm stops, if either the requested reduction is achieved, or no further reduction is possible.

To choose the value under **Triangle reduction / (%)** by either entering it in the field or moving the slider.

In the screenshots below, the effect of reducing the number of triangles by 25 % and 75 % is shown.



Edge Length

With the **Edge Length** mode, the number of triangles in the current mesh is reduced depending on two inputs: **Minimum Edge Length** and **Maximum Edge Length**.

The input given for **Minimum Edge Length** sets the length of edges to be removed. All edges shorter than this value are removed in the algorithm.

The input given for **Maximum Edge Length** sets the upper bound for the created edges which should not be exceeded.

The coarsening algorithm takes all edges smaller than the minimum edge length and combines them until no edge is smaller than the minimum edge length. The algorithm tries not to exceed the maximum edge length in the process.

Note: Nevertheless, it is not always possible to satisfy the given maximum edge length.

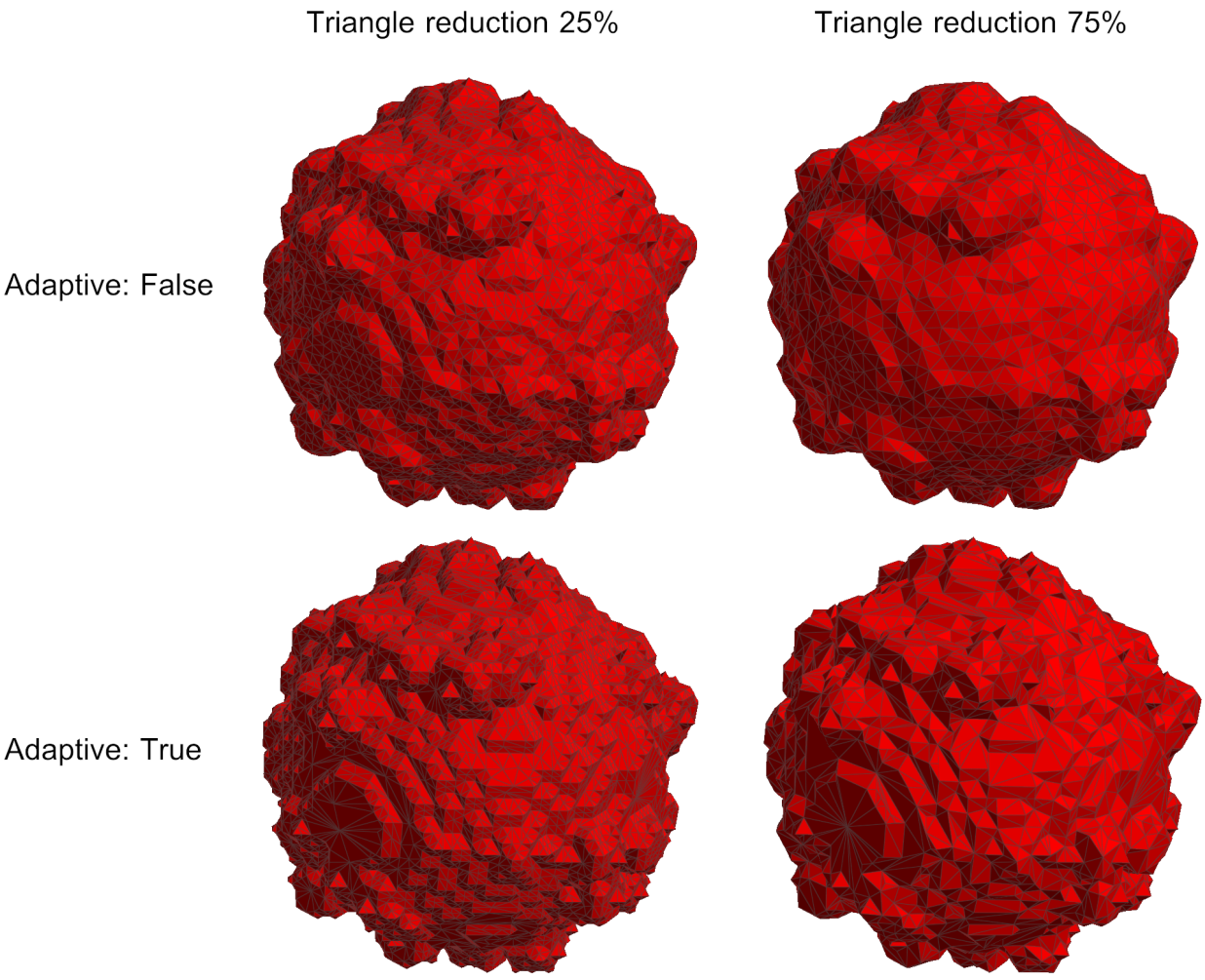
Target Number of Triangles

Target Number of Triangles reduces the number of triangles in the current mesh until the **Target Number of Triangles** is reached.

Adaptive

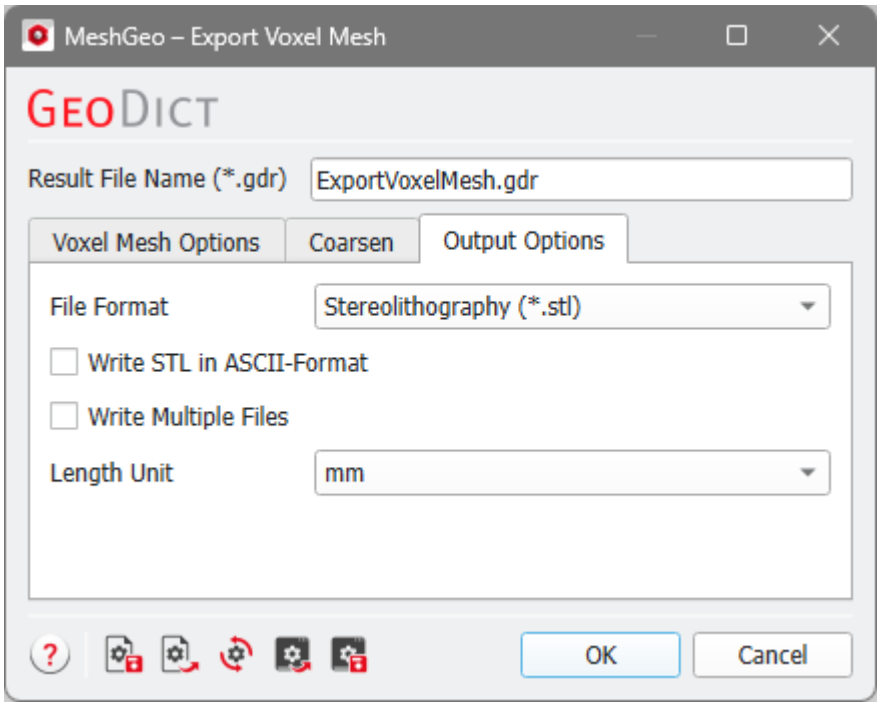
Adaptive is only available for the modes **Percentage Reduction of Triangles** and **Target Number of Triangles**. With **Adaptive**, the coarsening algorithm minimizes the deviation of the coarsened surface from the original surface. The **Adaptive** algorithm generates a mesh with a non-uniform triangle size and can lead to triangles with bad aspect ratios.

When the **Adaptive** mode is disabled, the coarsening algorithm works based on the edge lengths in the mesh.



1.1.3 Output Options

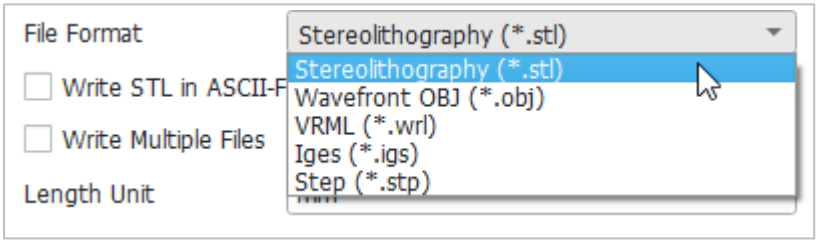
In the **Output Options** tab, select the file format and settings to save the mesh.



File Format

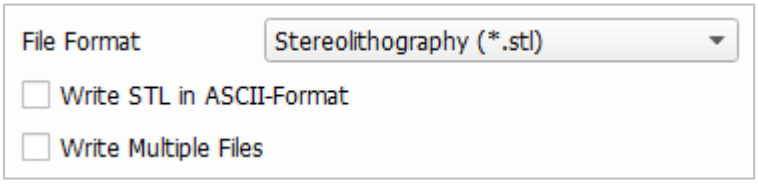
Select the **File Format** for the saved mesh.

The Stereolithography (*.stl), Wavefront OBJ (*.obj), VRML (*.wrl), Iges (*.igs), and Step (*.stp) formats can be exported natively by GeoDict. The third-party software CADLook, which was required for versions prior to GeoDict 2026 to export some of the file formats, is no longer necessary.



Write STL in ASCII-Format

This option is only available when the *.stl file format is chosen. When **Write STL in ASCII-Format** is selected, the file is saved as human readable ASCII file. Otherwise, the STL file is saved in binary format. The binary files are smaller and much faster to read and write. Therefore, the ASCII format should only be used if necessary.

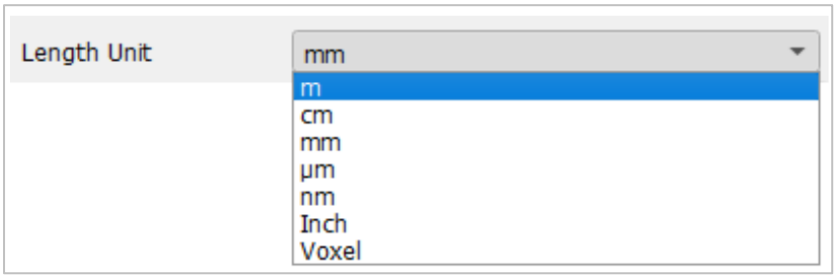


Write Multiple Files

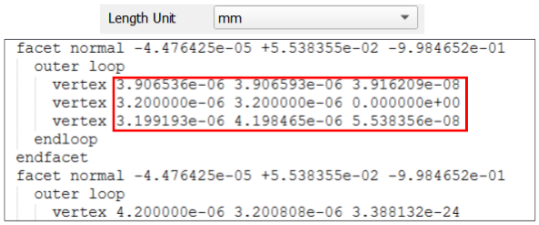
This option is also only available when the *.stl file format is chosen. When the STL format is chosen and multiple meshes are currently loaded, these meshes can be saved to different files if necessary. This option is enabled with **Write Multiple Files**. For the other file formats, multiple meshes are always saved to one file.

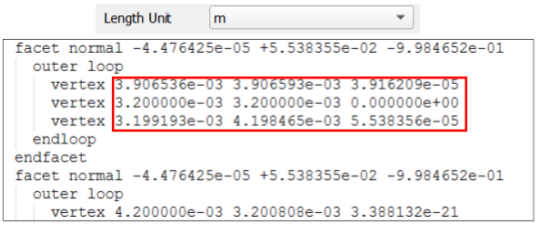
Length Unit

The unit in which the data is saved to the file can be selected with **Length Unit**. GeoDict can handle all these available units, but most external software cannot. Therefore, the unit should be chosen depending on the software with which the mesh is further processed.



In the STL format, the information about the used unit is not contained in the file. The files for different units differ only in the order of magnitude of the data. Therefore, the unit should be chosen carefully based on your needs.





1.2 Export Object Mesh

A mesh can be created based on the analytic objects in the structure with **Object Mesh**. Each object is meshed separately, therefore, if analytic objects overlap, the meshes for these objects might overlap, too. If it is important that these meshes do not overlap, use the **Multi Material** 9 mode with **Export Voxel Mesh** instead.

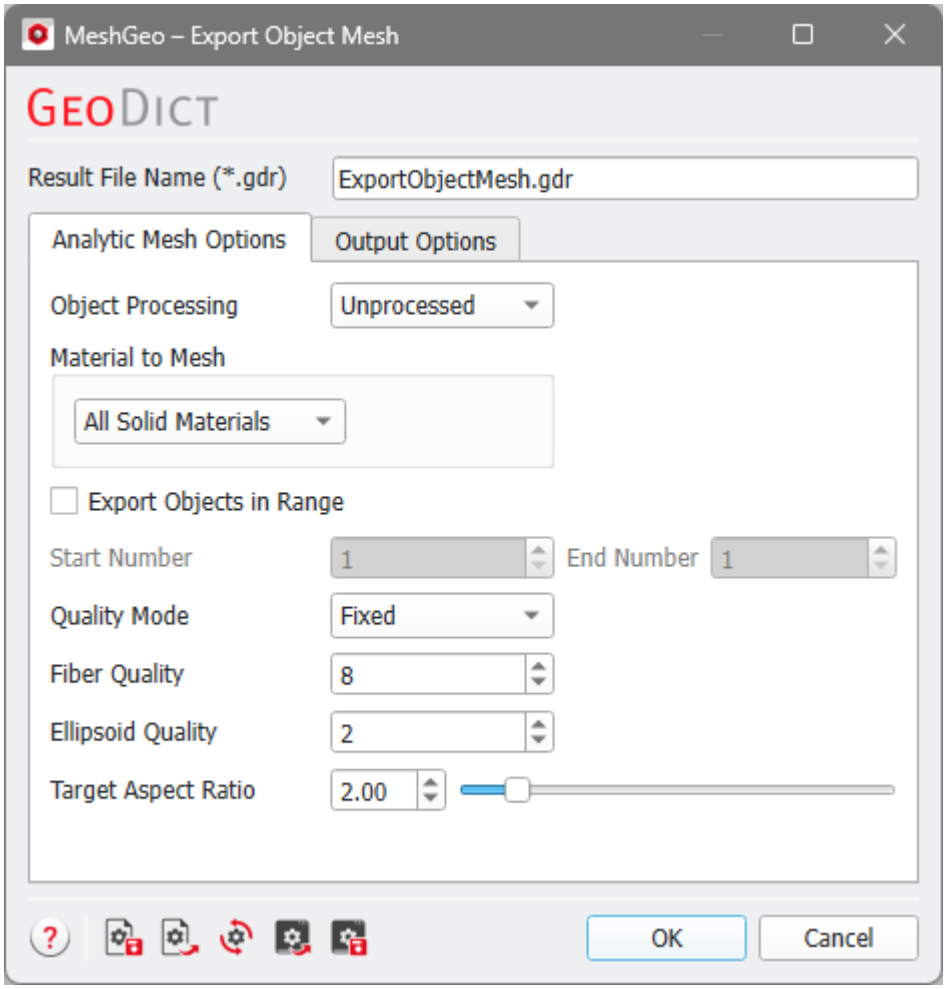
Analytic Mesh does not work for combined objects or intersected objects. Check out the GadGeo chapter for more information about these object types.

The Export Object Mesh options:

- Analytic Mesh Options 17
  Set the options for the creation of the mesh.
- Output Options 22
  Determine the output file format.

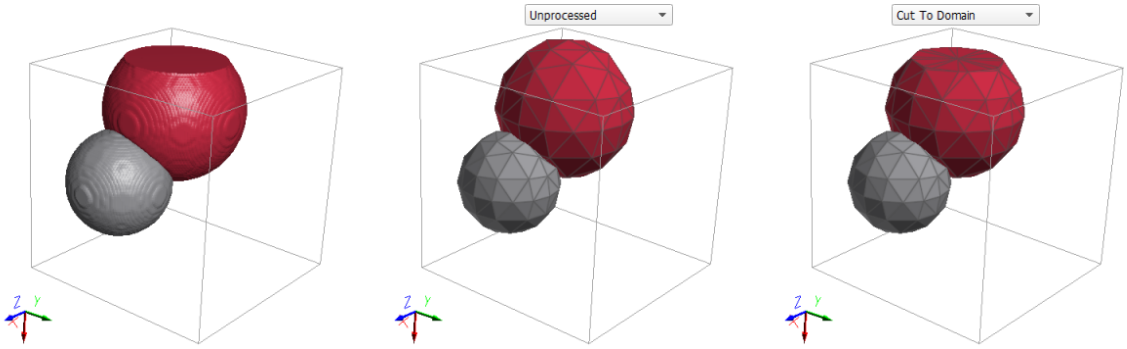
1.2.1 Analytic Mesh Options

The **Analytic Mesh Options** allow to create triangulations from the available GAD object information. The options include the **Object Processing** mode, **Material to Mesh**, **Export Objects in Range**, as well as the settings for defining the mesh quality.



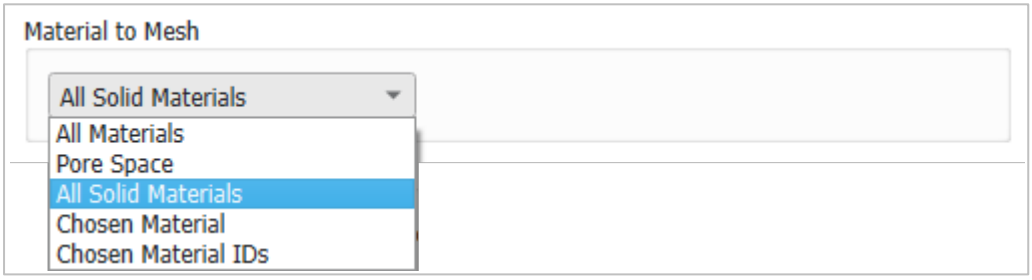
Object Processing

If **Object Processing** is set to **Unprocessed**, the complete objects are meshed, even if they lie not completely in the domain. With **Cut to Domain**, only the part of the object which is inside the domain is meshed.

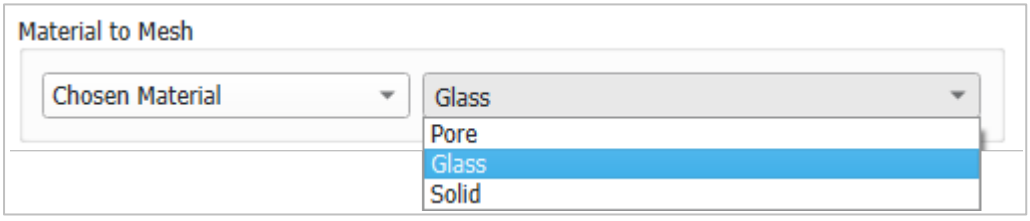


Material to Mesh

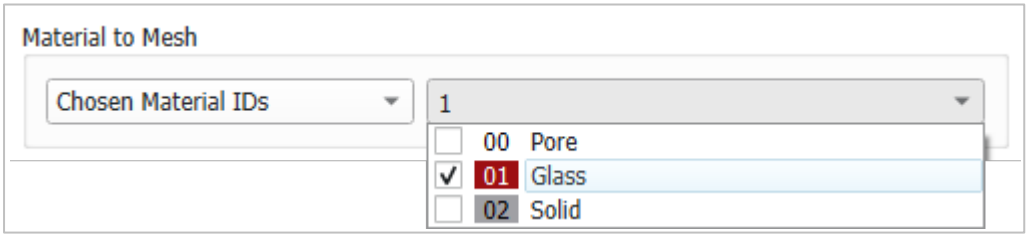
Select the Material IDs to be meshed with the options under **Material to Mesh**. With the first three options, either **All Materials**, the **Pore Space**, or **All Solid Materials** are meshed. **All Solid Materials** is the default, since this is the best choice for most use cases.



With **Chosen Material**, all Material IDs of the chosen material are meshed.



With **Chosen Material IDs**, individual Material IDs can be selected, which can have assigned different materials.

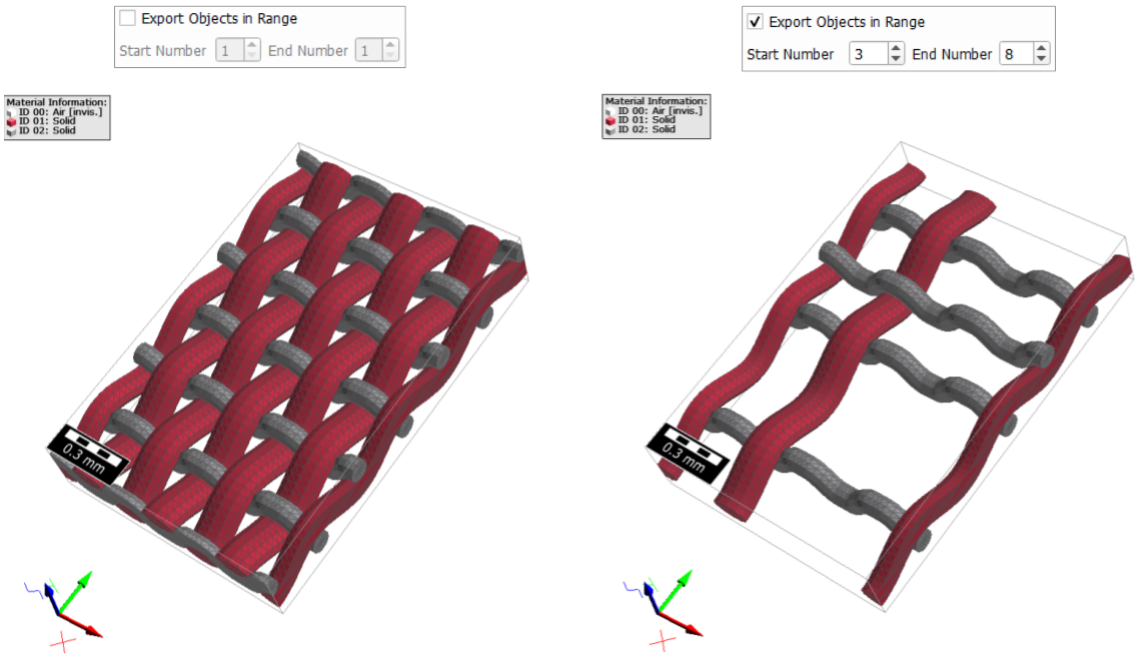


For **Analytic Mesh**, not all Material IDs in the structure can be used to create a mesh. For

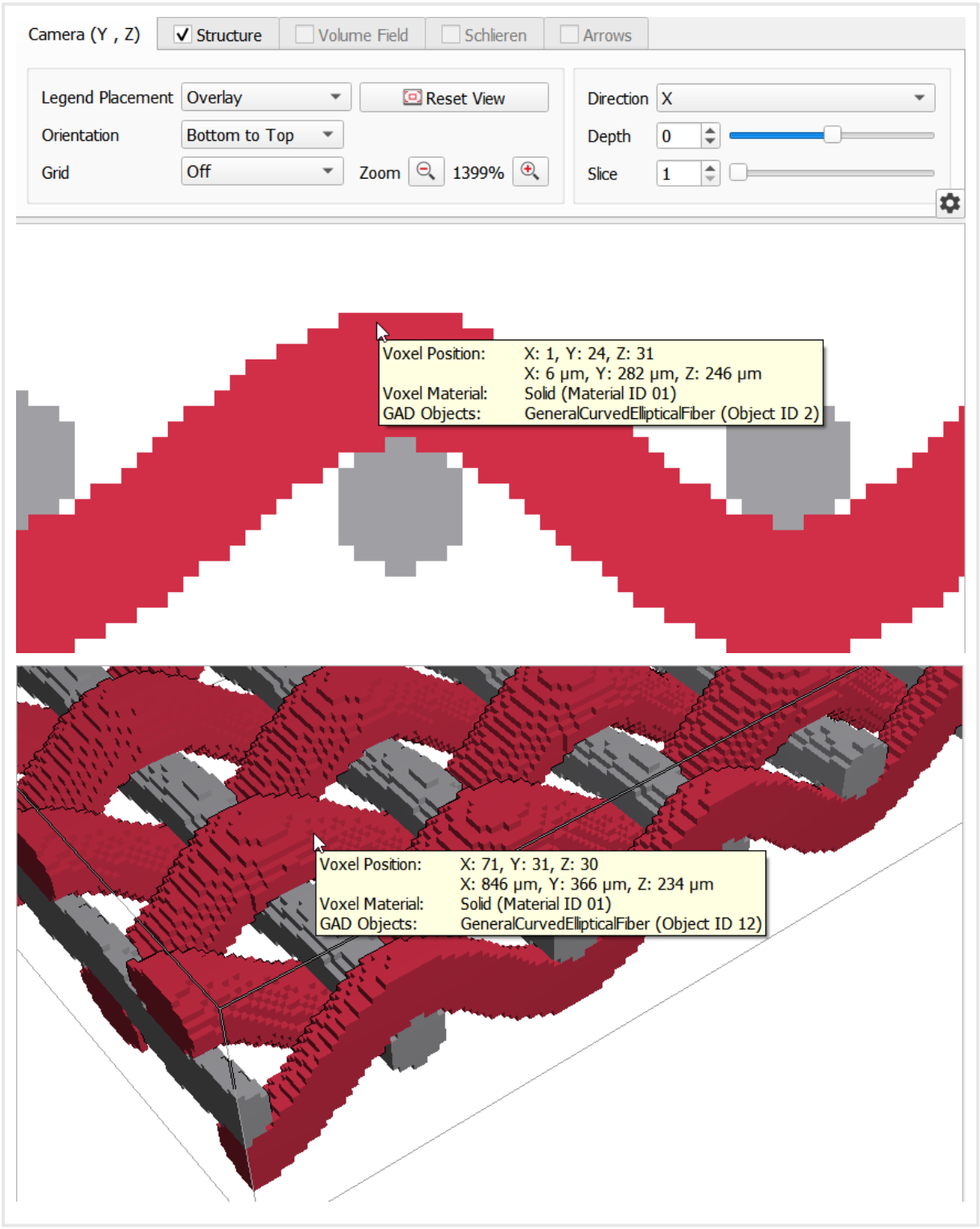
example, the pore space is not an object in GeoDict and therefore no analytic mesh can be created from it. Furthermore, the overlap between objects is not an analytic object itself and can therefore not be meshed. To create a mesh of, e.g., the pore space or the overlap regions, use the **Voxel Mesh** option instead.

Export Objects in Range

Each object in a structure has an object ID. If only some of these objects should be meshed, a range of objects can be selected with **Export Object in Range**. Alternatively, the structure can be edited so that it only contains the desired objects (for example, with GadGeo-Edit GAD Objects).

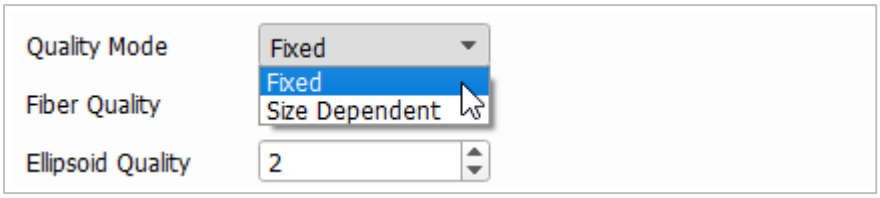


To find the object number of a given object, hover the mouse over the object in 2D or 3D view and check out the Tooltip.



Quality Mode

Objects in GeoDict can be meshed in different resolutions. The **Quality Mode** defines if this resolution is fixed or if it depends on the object size.



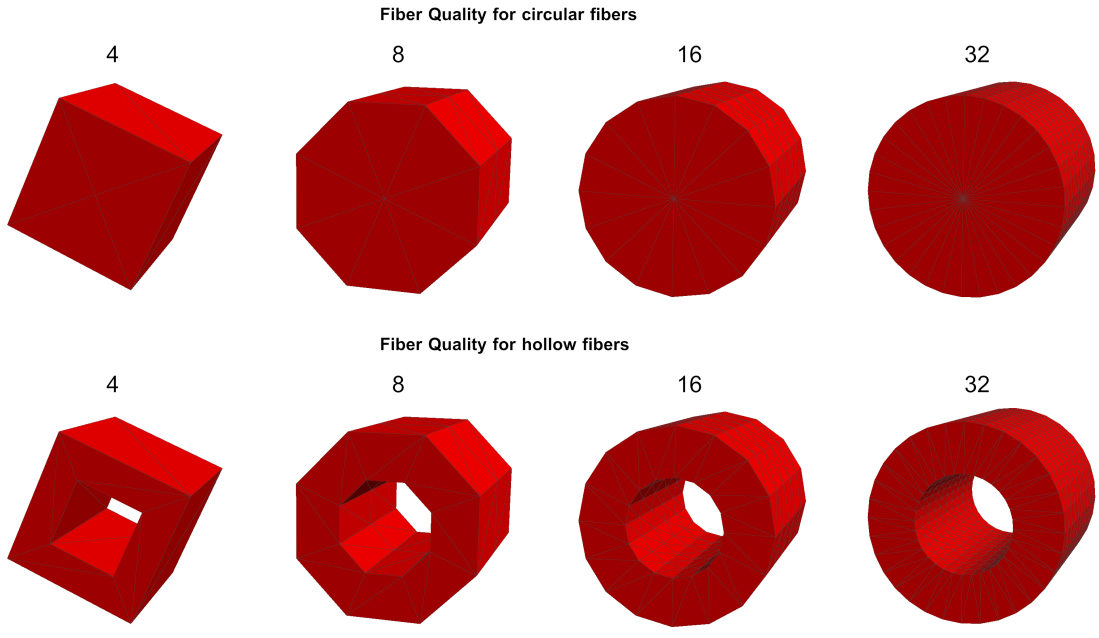
With **Quality Mode** set to **Fixed**, the **Fiber Quality** and **Ellipsoid Quality** are defined globally for the structure. When choosing **Size Dependent**, minimum and maximum values must be defined. This option is recommended if the size of the objects in the structure varies by a large amount.

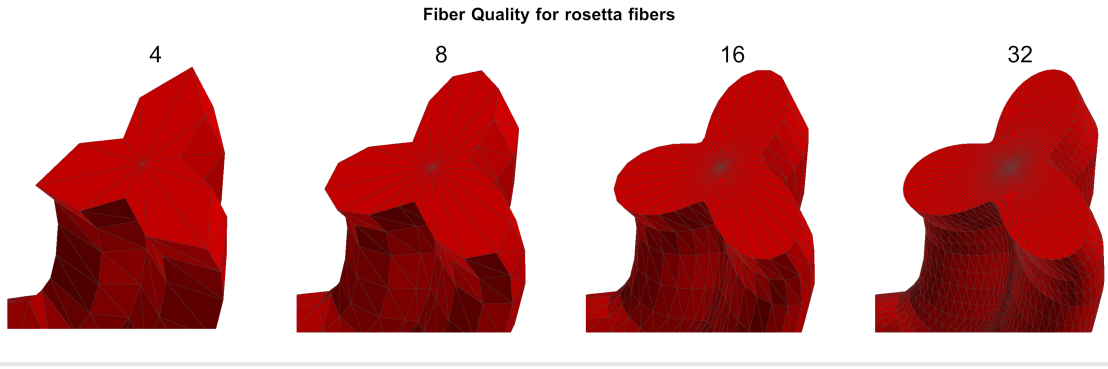
The quality can only be selected for fibers and ellipsoids (including spheres). For the other object types available in GeoDict, meshing is either trivial so that only one reasonable mesh exists (**Box**, **Triangle**, **Planar Polyhedron**, **Convex Polyhedron** type objects) or not possible in GeoDict (**Combined Object** and **Intersected Object**, where no analytical mesh definition exists).

Fiber Quality

Fiber Quality controls the resolution for fibers. The entered value determines the number of nodes in which the profile shape is resolved.

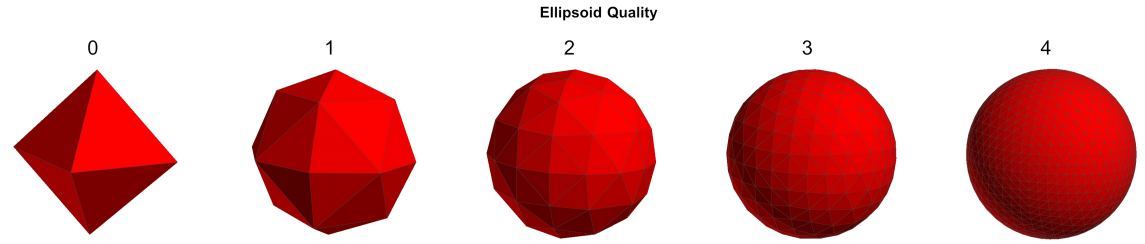
- For fibers with circular and ellipsoidal cross-section, the **Fiber Quality** corresponds to the number of nodes with which the cross-section of the fiber is discretized. This does also hold for fibers of hollow and cellulose types.
- For rectangular and angular fibers, the **Fiber Quality** has no effect on the discretization of the cross-section. This is because these fiber types are already perfectly resolved with one node per side.
- For Rosetta fibers, the **Fiber Quality** corresponds to the number of nodes per leaf.





Ellipsoid Quality

The **Ellipsoid Quality** determines the resolution of the mesh for spheres and ellipsoids. In contrast to the **Fiber Quality**, the **Ellipsoid Quality** is an exponential measure for the resolution. For the value 0, the shape of the sphere or ellipsoid is meshed as an octagon. With each increase of the parameter, each of the triangles is refined into four new triangles. Therefore, a meshed sphere with **Ellipsoid Quality = 0** contains 8 triangles and for **Ellipsoid Quality = 1**, it contains 32 = 8 x 4 triangles. For **Ellipsoid Quality = 2**, it contains already 256 = 8 x 4^2 and so on.

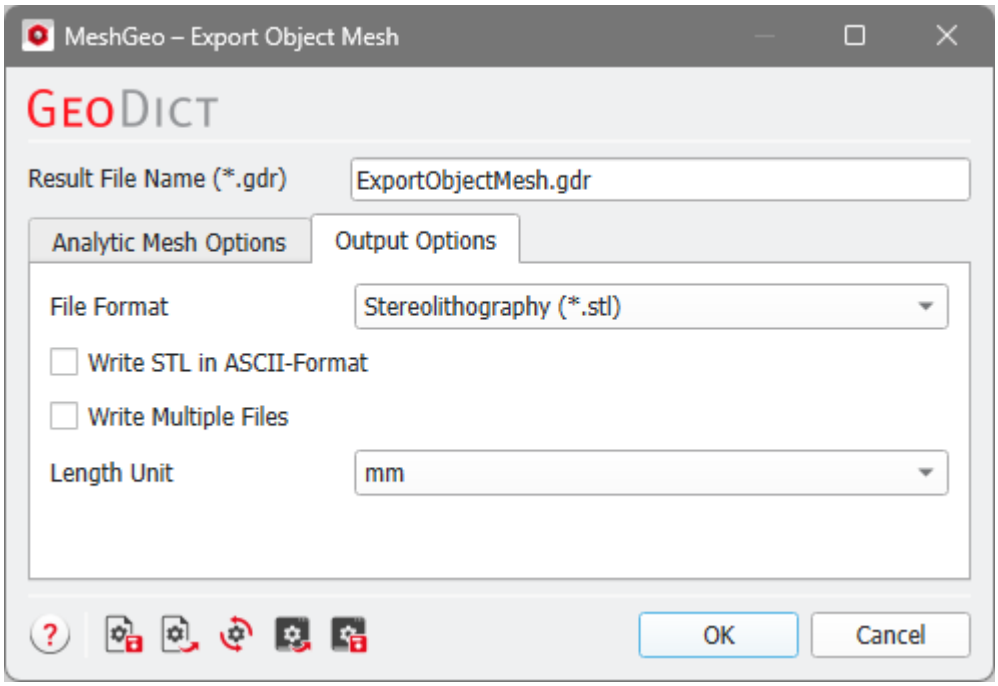


Target Aspect Ratio

The Target Aspect Ratio specifies the desired aspect ratio of the triangles in the surface triangulation. Lower values will create more triangles of higher quality.

1.2.2 Output Options

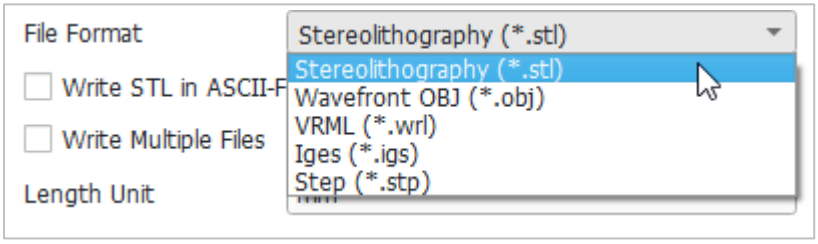
In the **Output Options** tab, select the file format and settings to save the mesh.



File Format

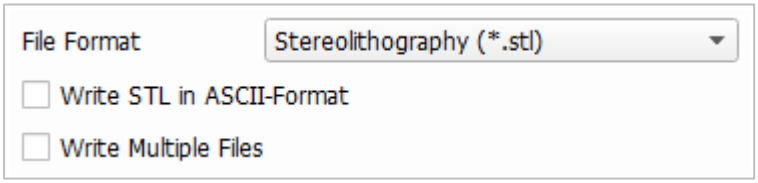
Select the **File Format** for the saved mesh.

The Stereolithography (*.stl), Wavefront OBJ (*.obj), VRML (*.wrl), Iges (*.igs), and Step (*.stp) formats can be exported natively by GeoDict. The third-party software CADLook, which was required for versions prior to GeoDict 2026 to export some of the file formats, is no longer necessary.



Write STL in ASCII-Format

This option is only available when the *.stl file format is chosen. When **Write STL in ASCII-Format** is selected, the file is saved as human readable ASCII file. Otherwise, the STL file is saved in binary format. The binary files are smaller and much faster to read and write. Therefore, the ASCII format should only be used if necessary.

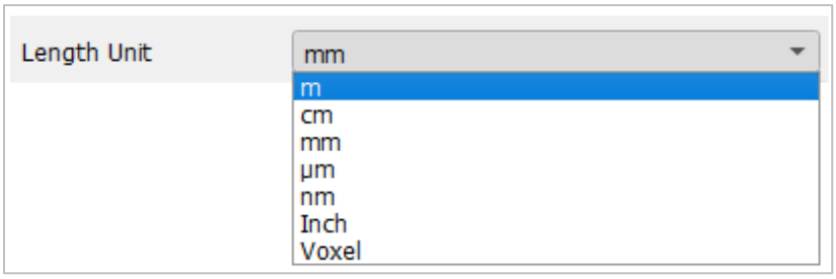


Write Multiple Files

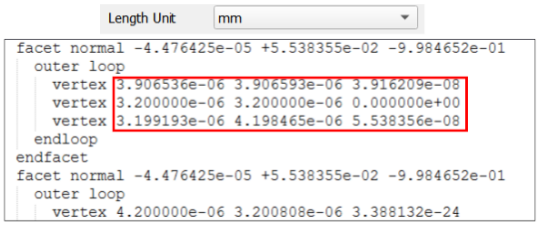
This option is also only available when the *.stl file format is chosen. When the STL format is chosen and multiple meshes are currently loaded, these meshes can be saved to different files if necessary. This option is enabled with **Write Multiple Files**. For the other file formats, multiple meshes are always saved to one file.

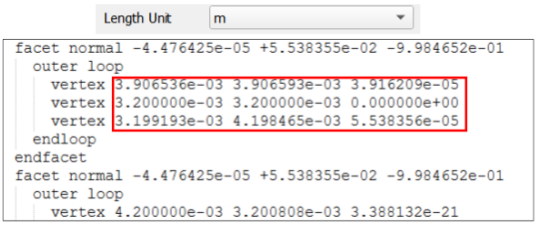
Length Unit

The unit in which the data is saved to the file can be selected with **Length Unit**. GeoDict can handle all these available units, but most external software cannot. Therefore, the unit should be chosen depending on the software with which the mesh is further processed.



In the STL format, the information about the used unit is not contained in the file. The files for different units differ only in the order of magnitude of the data. Therefore, the unit should be chosen carefully based on your needs.





1.3 Export Object CAD

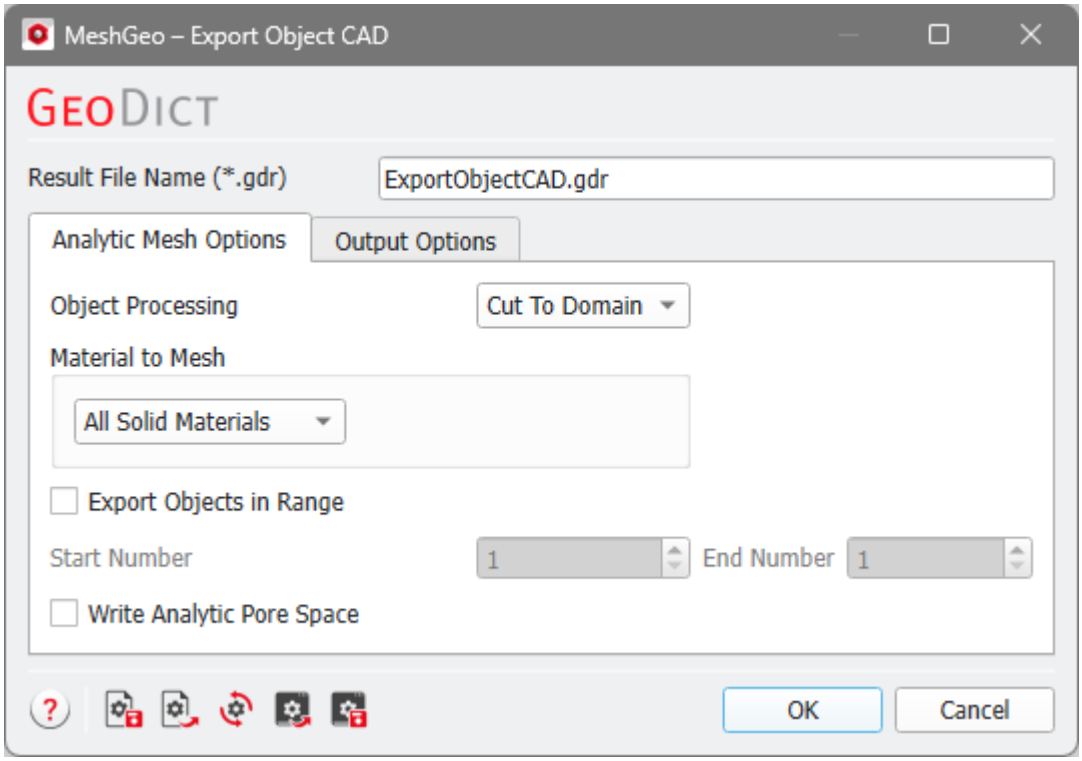
Export Object CAD exports the analytic geometry in its native CAD format. Unlike **Export Object Mesh**, which saves a triangulated surface, this command preserves the object's true CAD representation as precise primitives like cylinders and spheres.

The Export Object CAD options:

- Analytic Mesh Options 25
  Set the options for the creation of the CAD.
- Output Options 28
  Determine the output file format.

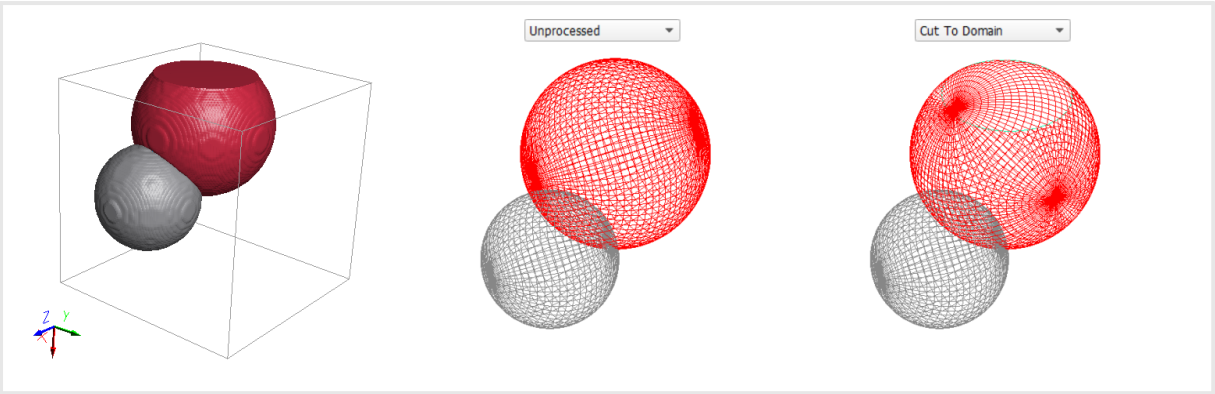
1.3.1 Analytic Mesh Options

The **Analytic Mesh Options** allow to choose for the available GAD objects for the **Object Processing** mode, **Material to Mesh**, **Export Objects in Range**, as well as whether to **Write Analytic Pore Space** or not.



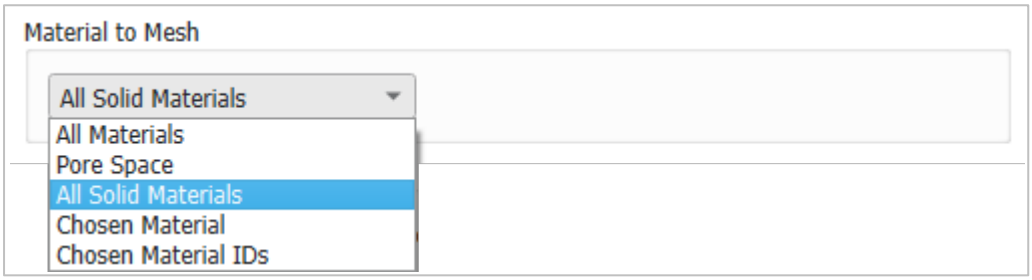
Object Processing

If **Object Processing** is set to **Unprocessed**, the complete objects are exported, even if they lie not completely in the domain. With **Cut to Domain**, only the part of the object which is inside the domain is exported. The differences can be viewed when you use some other CAD tools.

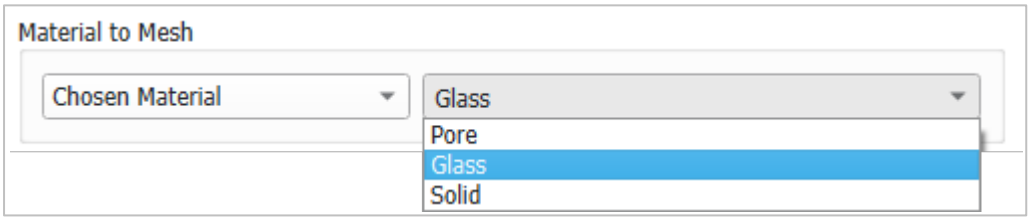


Material to Mesh

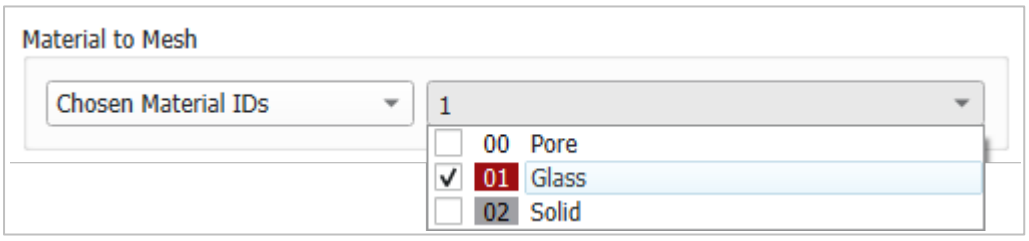
Select the Material IDs to be meshed with the options under **Material to Mesh**. With the first three options, either **All Materials**, the **Pore Space**, or **All Solid Materials** are meshed. **All Solid Materials** is the default, since this is the best choice for most use cases.



With **Chosen Material**, all Material IDs of the chosen material are meshed.



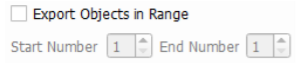
With **Chosen Material IDs**, individual Material IDs can be selected, which can have assigned different materials.

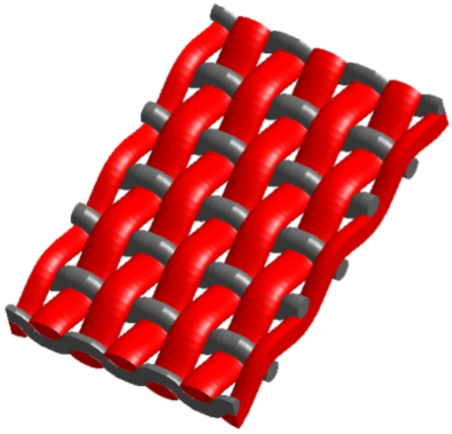


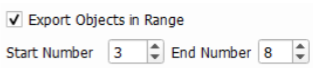
Export Objects in Range

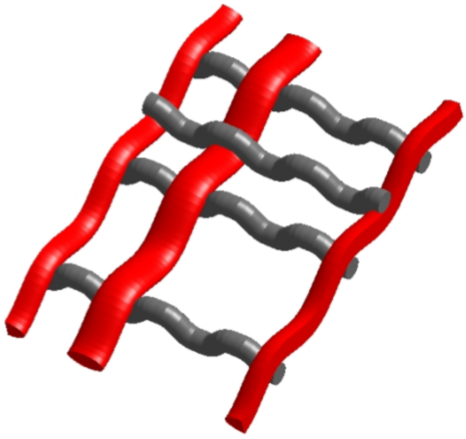
Each object in a structure has an object ID. If only some of these objects should be meshed, a range of objects can be selected with **Export Object in Range**. Alternatively,

the structure can be edited so that it only contains the desired objects (for example, with GadGeo-Edit GAD Objects).



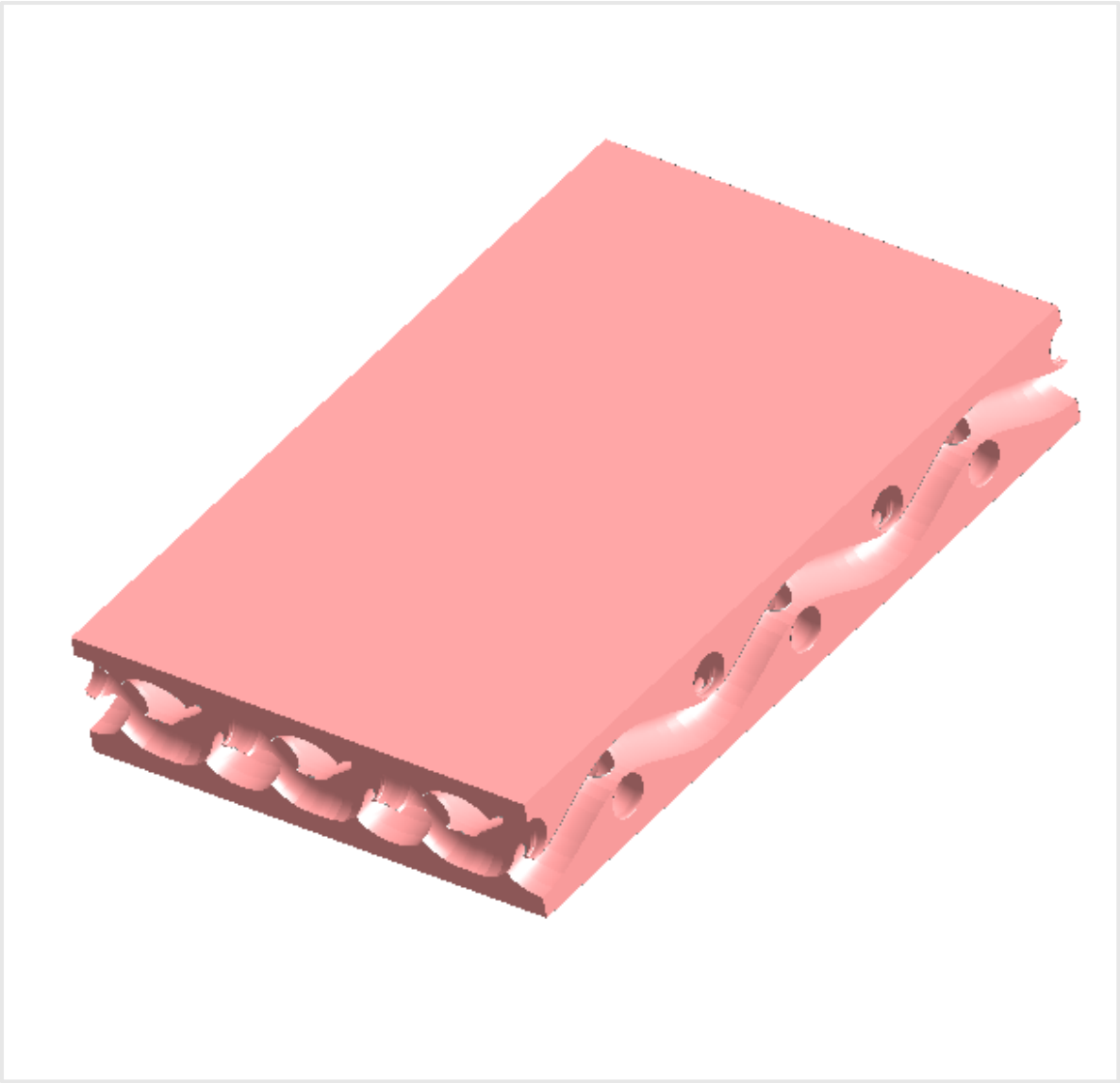






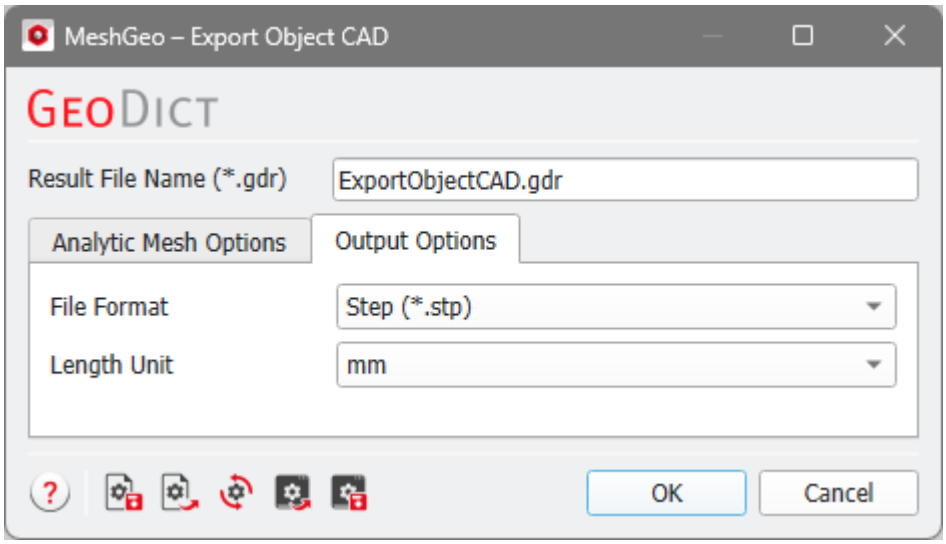
Write Analytic Pore Space

When **Write Analytic Pore Space** is checked, the pore space in the structure is written in the CAD file.



1.3.2 Output Options

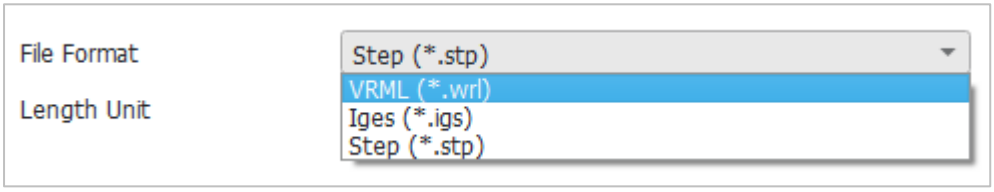
In the **Output Options** tab, select the file format and settings to save the mesh.



File Format

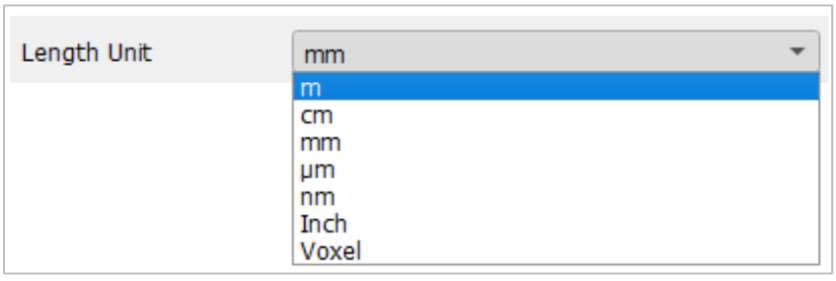
Select the **File Format** for the saved mesh.

The VRML (*.wrl), Iges (*.igs), and Step (*.stp) formats can be exported natively by GeoDict.



Length Unit

The unit in which the data is saved to the file can be selected with **Length Unit**. GeoDict can handle all these available units, but most external software cannot. Therefore, the unit should be chosen depending on the software with which the mesh is further processed.



Citation: Math2Market GmbH, 2026: GeoDict 2026 User Guide, ExportGeo-CAD, Math2Market GmbH, Germany, doi.org/10.30423/userguide.geodict

MATH 2 MARKET

Math2Market GmbH
Richard-Wagner-Str. 1, 67655 Kaiserslautern / Germany
www.math2market.com