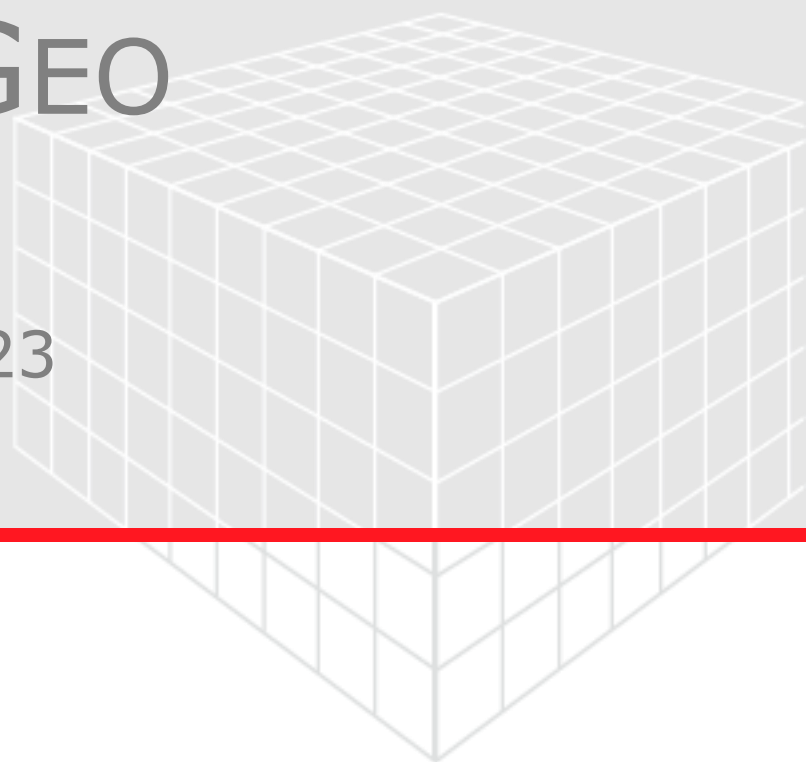


# EXPORTGEO-CAD WITH MESHGEO

User Guide

GeoDict release 2023

Published: August 18, 2023



# GEO DICT

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

© Math2Market GmbH 2023

Citation:

Liping Cheng, Sebastian Rief, Barbara Planas. GeoDict 2023 User Guide. ExportGeo-CAD with MeshGeo handbook. Math2Market GmbH, Germany, [doi.org/10.30423/userguide.geodict](https://doi.org/10.30423/userguide.geodict)

All rights reserved. It is not permitted to reproduce the book or parts thereof in any form by photocopy, microfilm or other methods or to transfer it into a language suitable for machines, in particular data processing systems, without the express permission of the publisher. The same applies to the right of public reproduction.

The handbooks in the User Guide series of Math2Market GmbH can be obtained from:

Math2Market GmbH  
Richard-Wagner-Strasse 1  
67655 Kaiserslautern  
Germany

Phone: +49 631 205 605 0  
Fax: +49 631 205 605 99  
Email: [info@math2market.de](mailto:info@math2market.de)  
Web: [www.math2market.de](http://www.math2market.de)

VISUALIZE SURFACE MESHES IN GEODICT	3
CREATE MESH	4
VOXEL MESH	5
OBJECT MESH	11
PARTICLES MESH	15
TRAJECTORIES MESH	16
STREAMLINES MESH	18
SAVE MESH	19
SMOOTH MESH	22
COARSEN MESH	25
REPAIR MESH	28
REMESH MESH	30
VOXELIZE MESH	32
SPLIT MESHES INTO CONNECTED COMPONENTS	33
DELETE MESH	35
REASSIGN MESH MATERIAL ID	36
SHIFT MESH	37
SCALE MESH	38
ROTATE MESH	40

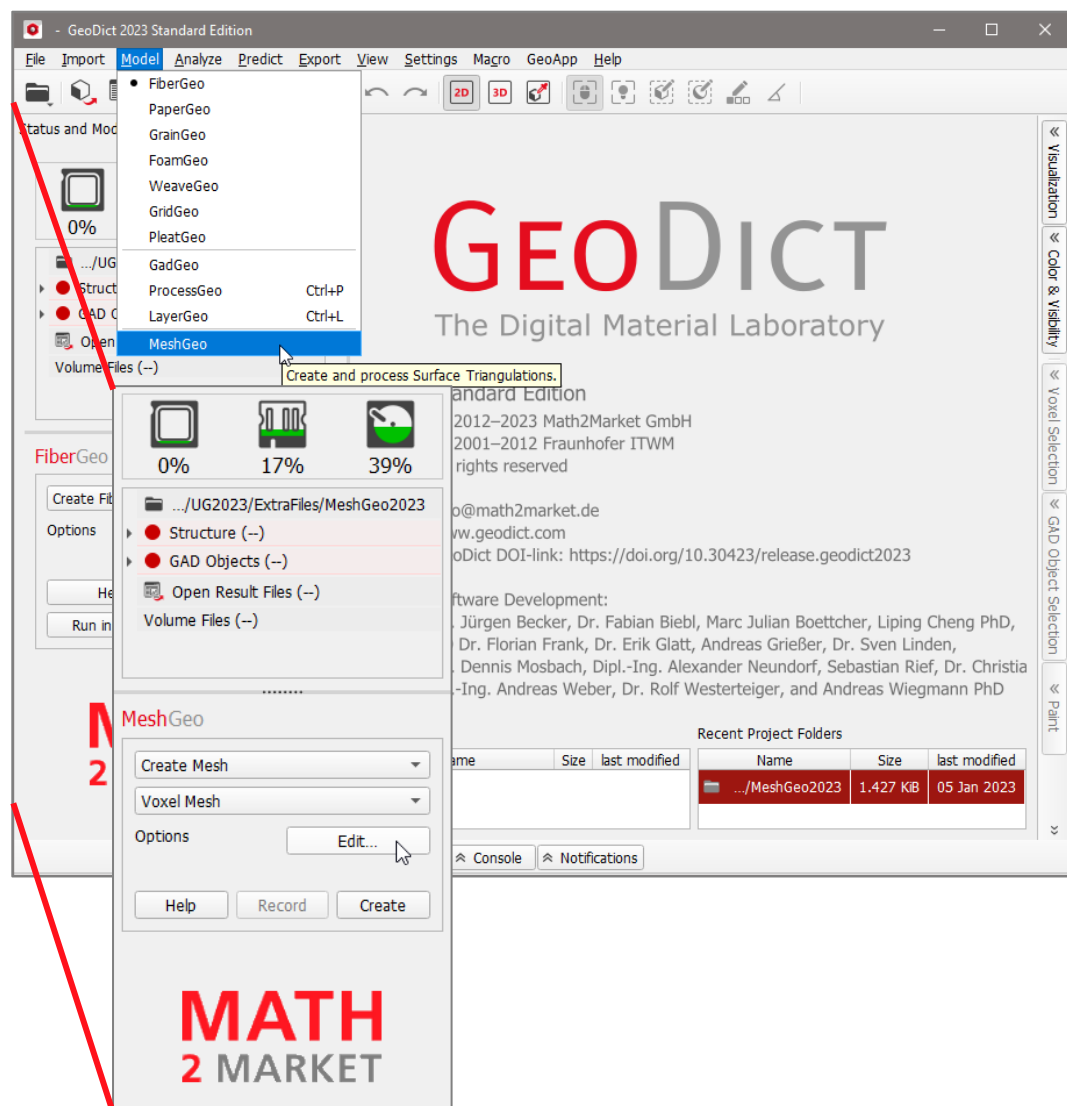
# MESHGEO / EXPORTGEO-CAD

With **MeshGeo** and **ExportGeo-CAD**, structures in **GeoDict** formats can be converted to a surface mesh. Since **GeoDict 2021**, it is also possible to create meshes directly from grey value images, index images and volume fields. These options add many new possibilities: For example, it is now possible to import  $\mu$ CT images and convert them to smooth meshes with sub-voxel precision. Furthermore, creating triangulation from **GeoDict** index images allows the transfer of the object information from **GeoDict**'s structure generators to external software.

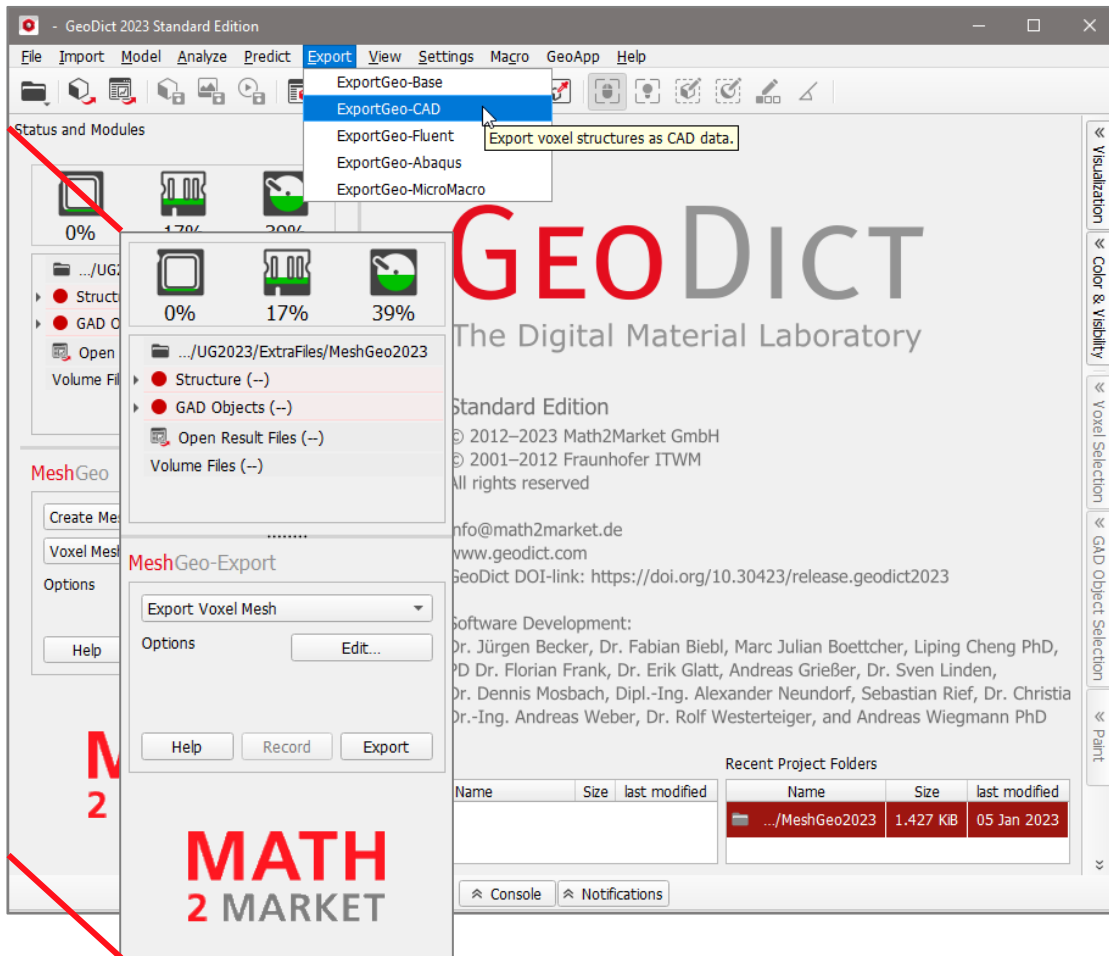
- **MeshGeo** keeps the mesh in memory after creation and allows subsequent smoothing and coarsening operations. For this, **MeshGeo** contains the additional options, **Smooth Mesh** and **Coarsen Mesh**. Afterwards, the mesh can be saved in different file formats with **Save Mesh**.
- **ExportGeo-CAD** saves the created mesh directly to the hard disk. Therefore, it is not possible to edit the mesh after creation.

Since both modules work in a similar way, only **MeshGeo** is explained in this handbook where it matches the functionality of **ExportGeo-CAD**.

To access **MeshGeo**, select **Model** → **MeshGeo** in the menu bar:



To access **ExportGeo-CAD**, select **Export** → **ExportGeo-CAD** in the menu bar:



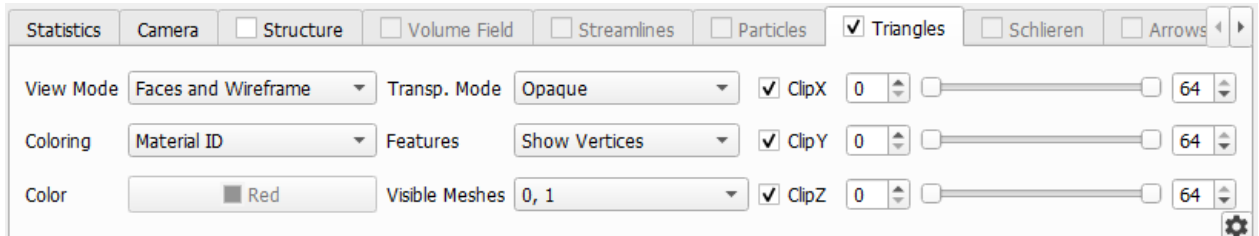
In the **MeshGeo** or **ExportGeo-CAD** section, to the left of the Visualization area, a pull-down menu displays the choices for the meshing options. The options available for **Create Mesh** are the same for both modules.

Additionally, **MeshGeo** contains options to further edit and save the mesh. For **ExportGeo-CAD**, the mesh is directly saved after the creation. The available options to save the mesh are the same for both modules and explained together in pages [10](#) ff.

## VISUALIZE SURFACE MESHES IN GEODICT

When a mesh is currently loaded in **GeoDict** and displayed in the Visualization area, the **Triangles** tab becomes available in the Visualization panel, above the Visualization area.

In this tab, various settings for the visualization of the mesh can be selected. These settings are explained in detail in the [ImportGeo-CAD](#) handbook of this User Guide.



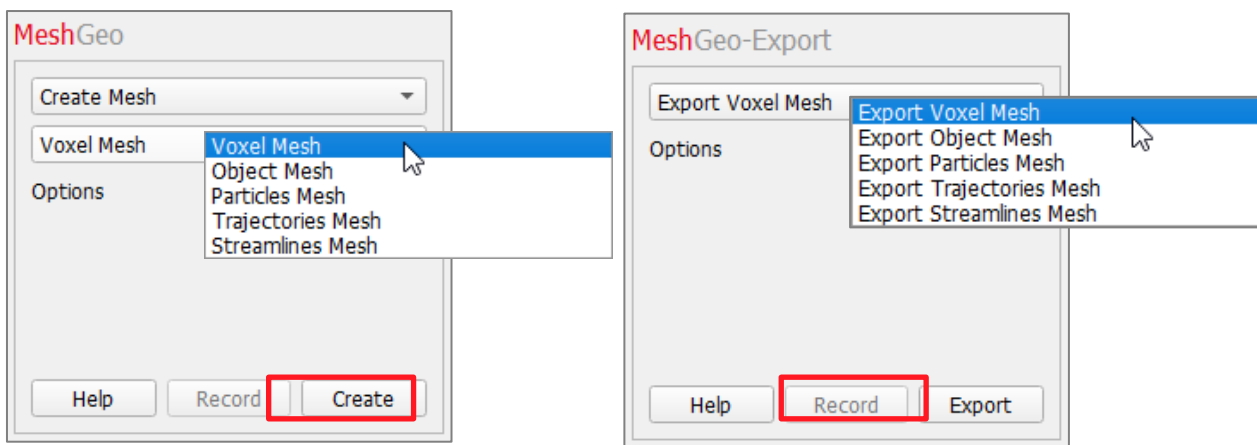
## CREATE MESH

Select **Create Mesh** in **MeshGeo** or **Export Voxel Mesh** in **ExportGeo-CAD** and specify the type of mesh from the pull-down menu. Meshes of a structure in **GeoDict** can be created based on the voxel information (**Voxel Mesh**). This voxel information can be a structure, a grey value image, an index image, or a volume field.

Alternatively, a mesh can be created based on the analytic objects in the structure (**Object Mesh**).

Additionally, it is also possible to mesh particles (**Particles Mesh**) and their trajectories (**Trajectories Mesh**) from results obtained with **AddiDict** or **FilterDict** or streamlines (**Streamlines Mesh**) from results obtained with **FlowDict**, **AddiDict** or **FilterDict**.

The parameters for the selected mesh type can then be defined by clicking the **Edit...** button.



When all settings have been chosen, click **Run** at the bottom of the **MeshGeo** or **Export** at the bottom of the **ExportGeo-CAD** section to create the mesh.

## VOXEL MESH

For voxel meshes in MeshGeo, two tabs are available in the **Create Voxel Mesh Options** dialog (**Voxel Mesh Options** and **Coarsen**).

A third tab, **Output Options** is only available in ExportGeo-CAD since the mesh is directly saved after creation.

It contains the same options as **Save Mesh** in MeshGeo. See page [10](#) for further information.



## VOXEL MESH OPTIONS

The **Voxel Mesh** option allows to create triangulations from the available voxel information. The kind of voxel information is selected with the **Volume** pull-down menu: It can be a GeoDict structure (this is the default), a grey value image, an index image, or a volume field. Additionally, the **Mode** defines how the volume is meshed. The choices for **Volume** and **Mode** change the other available options.

### VOLUME

Different types of **Volume** of structure information can be meshed in **Create Mesh** → **Voxel Mesh**. The available options are explained below.

#### Current Structure

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

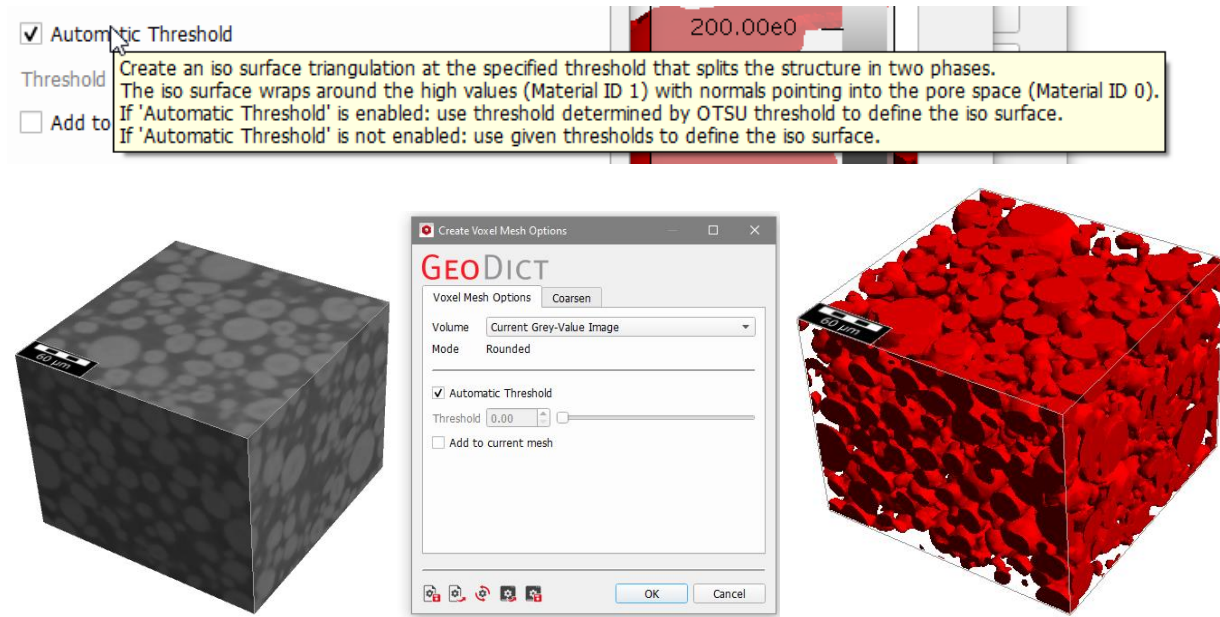
#### Current Grey-Value Image

Instead of first converting a grey-value image to a voxel structure (with **ImportGeo - Vol**), it is now directly possible to directly mesh the **Current Grey-Value Image**. In this case, the **Mode** is fixed to **Rounded**. Based on the given **Threshold**, an iso surface is created that splits the structure in two phases. The normal of the iso surface point into the pore space. If **Automatic Threshold** is selected, the OTSU algorithm is used to determine the threshold.



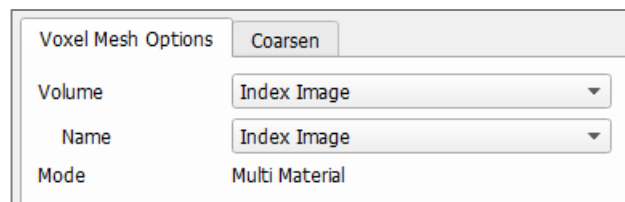
## MeshGeo and ExportGeo-CAD: convert structures to surface triangulations

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

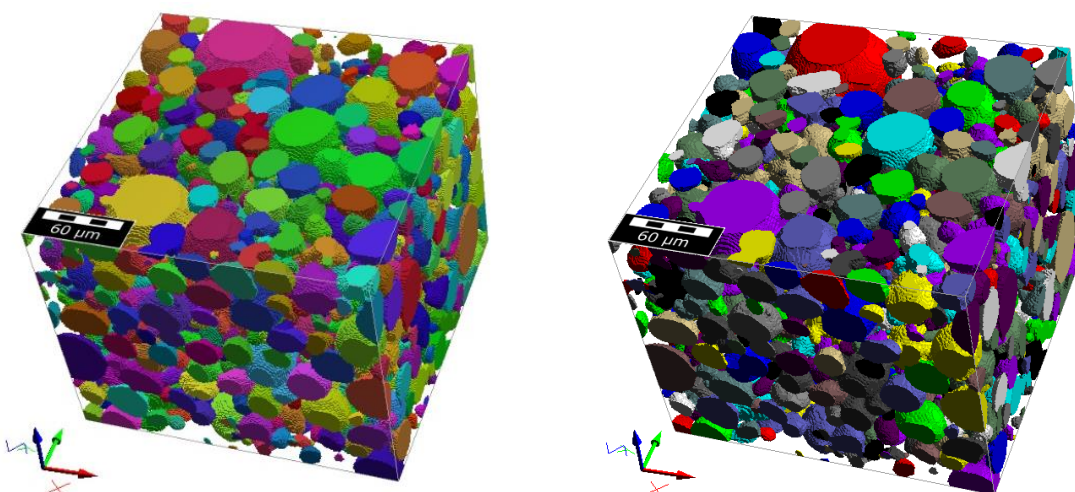


### Index Image

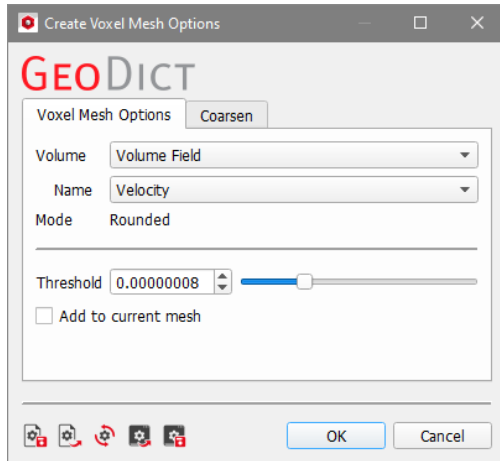
By choosing an **Index Image** as Volume, it is possible to use the object indexes to create **Multi Material** triangulations. Analogously, the object indexes from **GrainFind** or **FiberFind** results can be used in the same way. Each object with a given index is converted to its own mesh, and the 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.



In the figure below, to the left an index image from a **GrainFind** result is shown. To the right, you can see the mesh calculated from that index image.

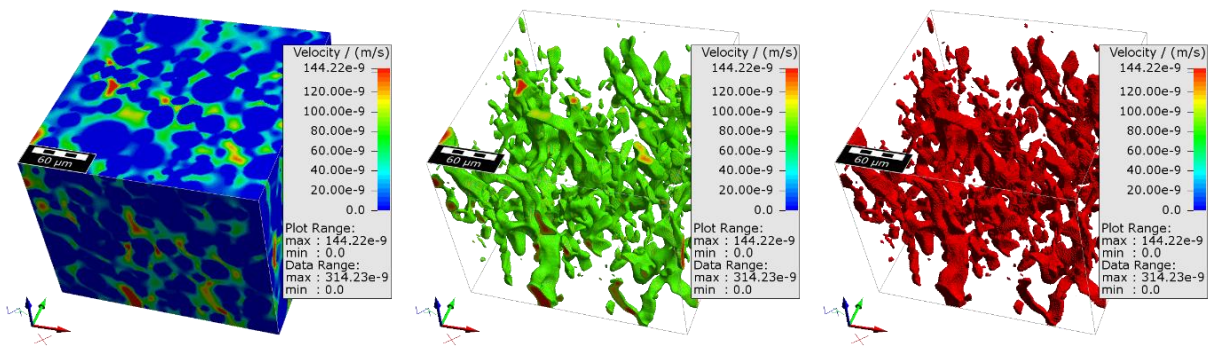


Volume Field



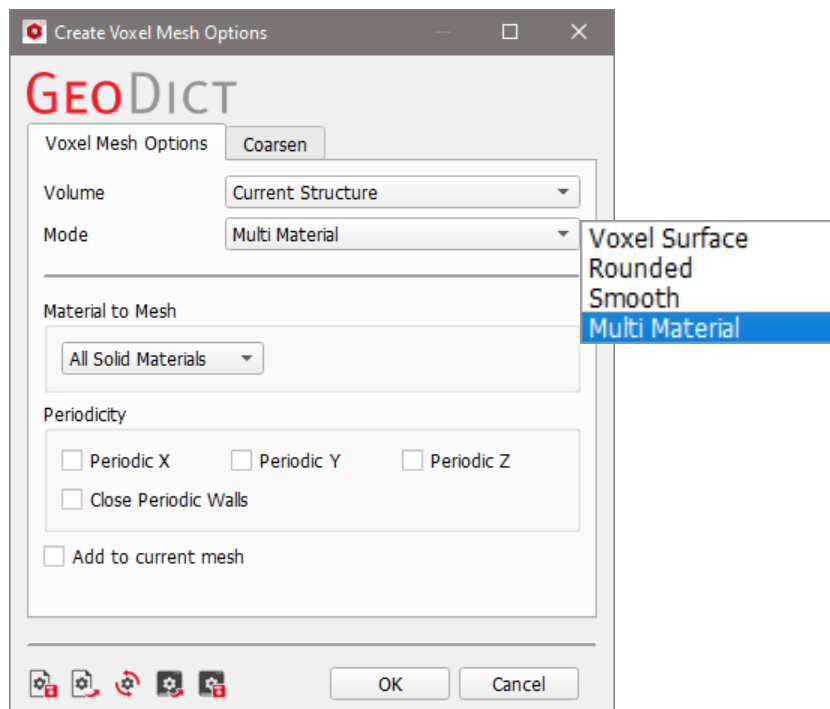
With the Volume Field option, triangulations can be created from arbitrary GeoDict results. In the example below, a flow field from a FlowDict simulation is converted to a mesh. The **Volume Field** option works analogously to the **Current Grey-Value Image** option: It creates an iso-surface mesh based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

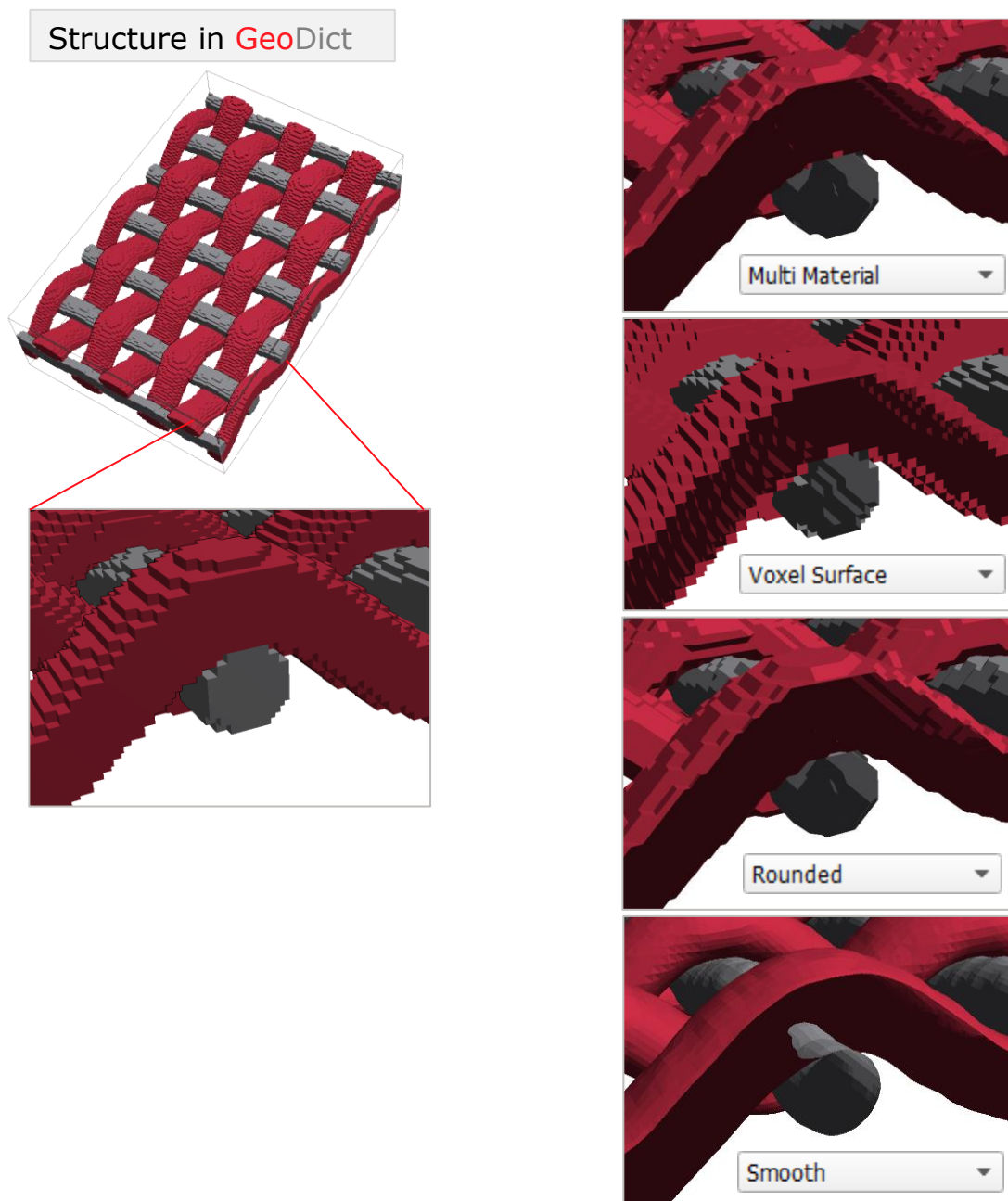
Four **Modes** can be used to export voxel data: **Voxel Surface**, **Rounded**, **Smooth**, or **Multi Material**. **Multi Material** is the default. Which options are available depends on the selected Volume: For example, if the **Current Grey-Value Image** is selected, only **Rounded** is available.



**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. In general, it is not recommended to use **Voxel Surface** since the **Multi-Material** mode leads to better results.

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.

**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).



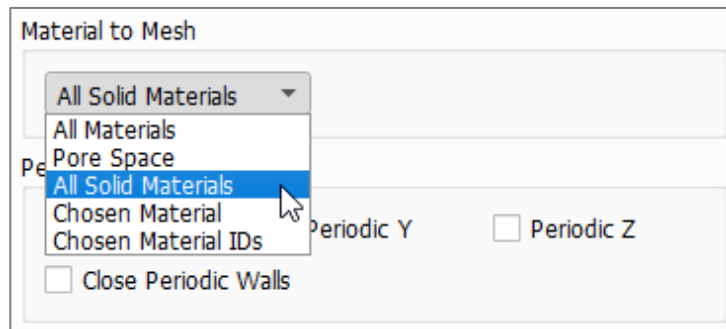
With the **Multi Material** option, the meshes for the different materials do not overlap. They have perfect contact at the points where different materials touch. When all materials are exported, a partition of unity is created. This means that each point in the domain is enclosed in exactly one surface.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

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

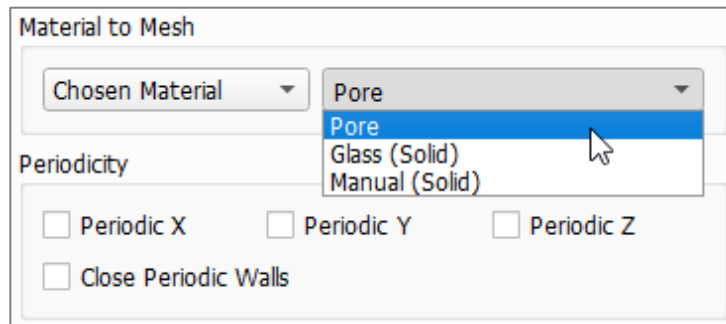
**MATERIAL TO MESH**

Select the material IDs to be meshed with the options under **Material to Analyze**. With the first three options, either **All Materials**, the **Pore Space** or **All Solid Materials** are meshed.

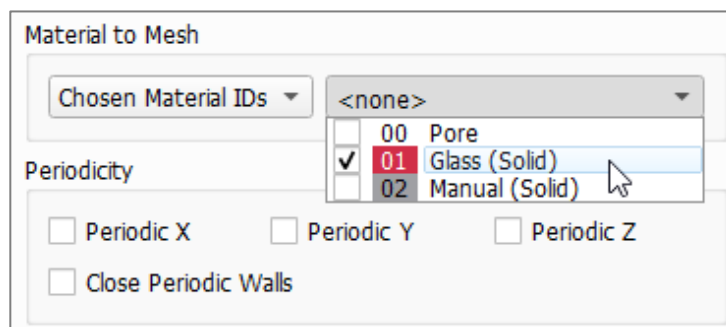
**All Solid Materials** is the default, since this is the best choice for most use cases.



With **Chosen Material**, all material IDs for the given material are selected.



With **Chosen Material IDs**, individual material IDs can be selected.



### PERIODICITY

---

When **Periodicity** is selected, a periodic mesh is created in the chosen direction. This option should only be used if the original structure is periodic. Otherwise, symmetric boundary conditions are applied. This option is only available for the **Multi Material** mode.

### CLOSE PERIODIC WALLS

---

When **Periodicity** is selected for a direction, the mesh walls are by default open at the boundaries. With **Close Periodic Cell Walls**, these periodic walls are closed. This option is only available for the **Multi Material** mode.

### AVOID REPEATING EDGES

---

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. This option is only available for **Voxel Surface** mode.

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

### ADD TO CURRENT MESH

---

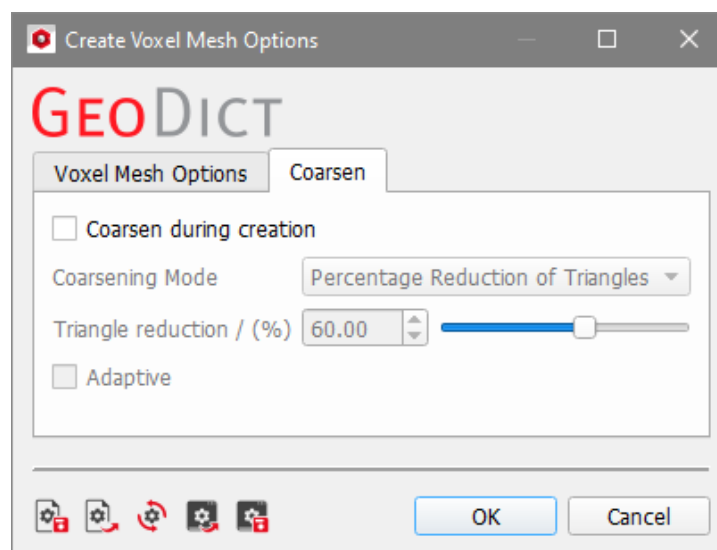
**Add to current** mesh is only available for MeshGeo. When this option is selected and a mesh is already in memory, the new mesh is added to that mesh. Multiple meshes can then be edited together and be saved to a single file.

See [Smooth Mesh](#) in page [22](#) and [Coarsen Mesh](#) in page [25](#).

### COARSEN

---

Coarsening can either be applied during mesh creation or afterwards. The settings are the same, but the results may differ – especially if the **Adaptive** mode is used. For further information, see page [25](#).

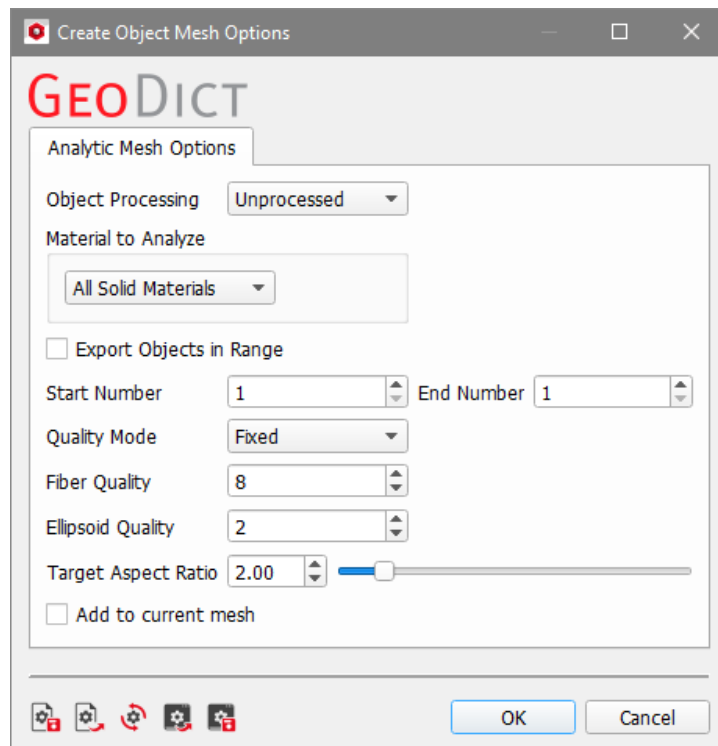




## OBJECT MESH

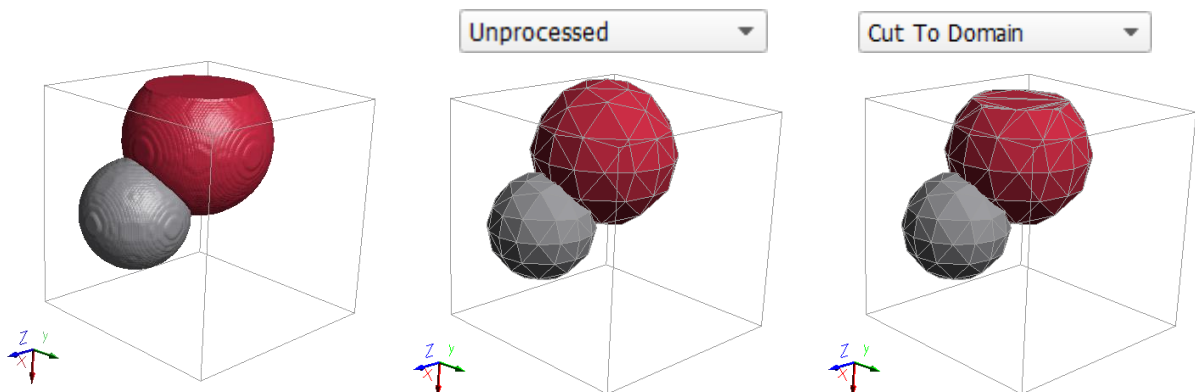
To export the objects in the structure into a mesh, select **Analytic Mesh** from the pull-down menu. Objects in **GeoDict** can overlap. With **Analytic Mesh**, each object is meshed separately, therefore the meshes for the different objects can overlap, too. If it is important that these meshes do not overlap, use the **Multi Material** mode (see page 9) instead.

**Analytic Mesh** does not work for combined objects or intersected objects. Check out the [GadGeo](#) handbook of this User Guide for more information about these object types.



## 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.

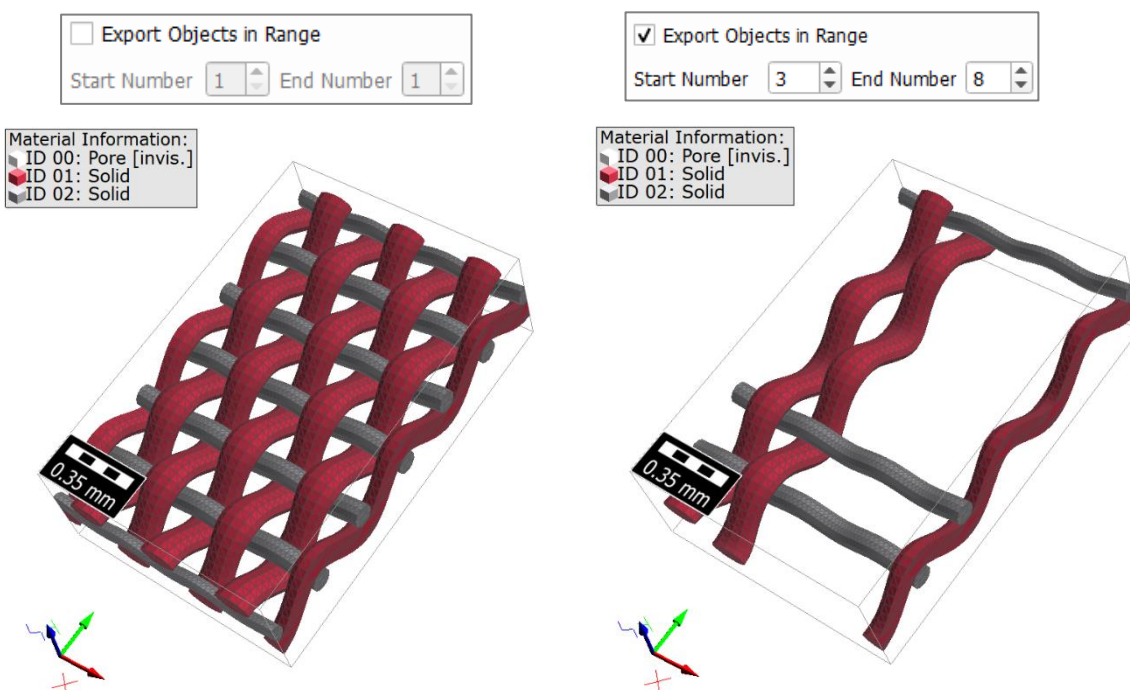


### MATERIAL TO ANALYZE

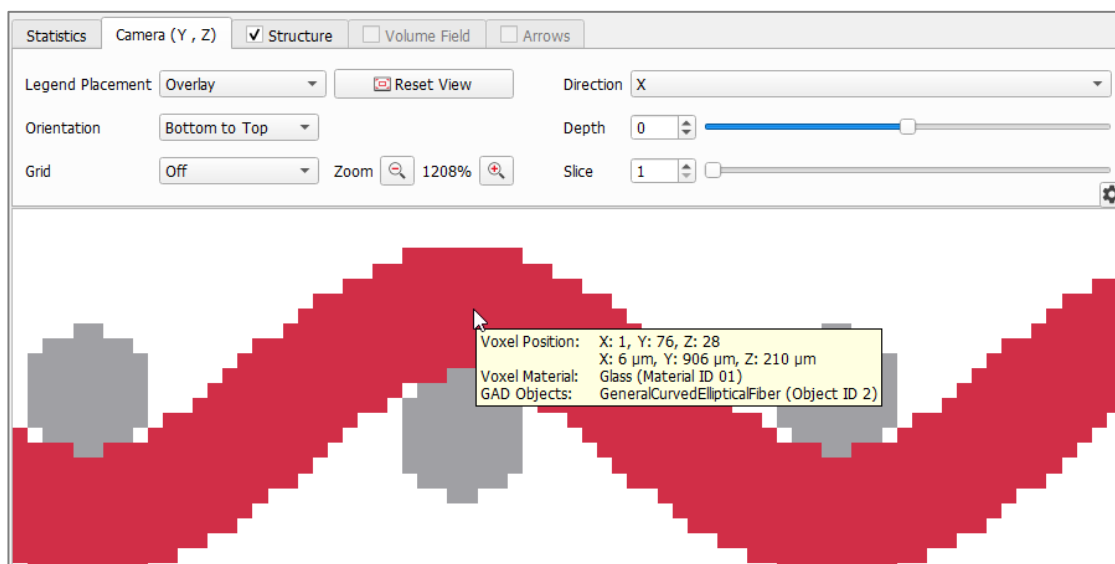
The choices for **Material to Analyze** are explained on page 9. For **Analytic Mesh**, not all material IDs in the structure can be used to create a mesh. For example, the pore space is not an object in **GeoDict** and therefore no analytic mesh can be created from it. Furthermore, the overlap between 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.

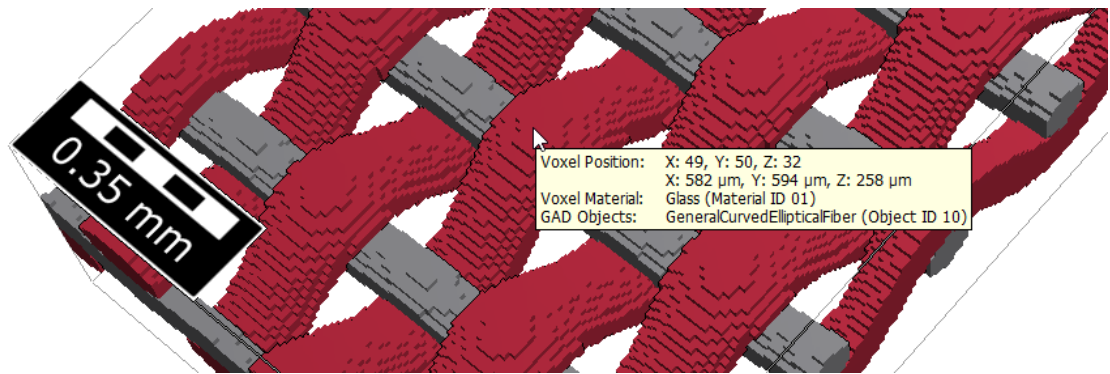
### EXPORT OBJECTS IN RANGE

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**). See the [GadGeo](#) handbook of this User Guide for further information).



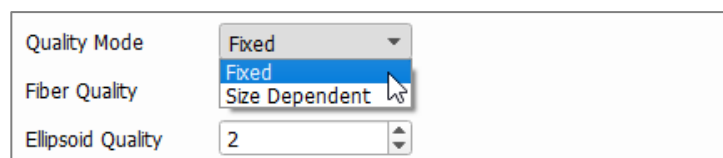
To find the object number of a given object, hover the mouse over the object in 2D or 3D view and check out the ToolTip.





## 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.



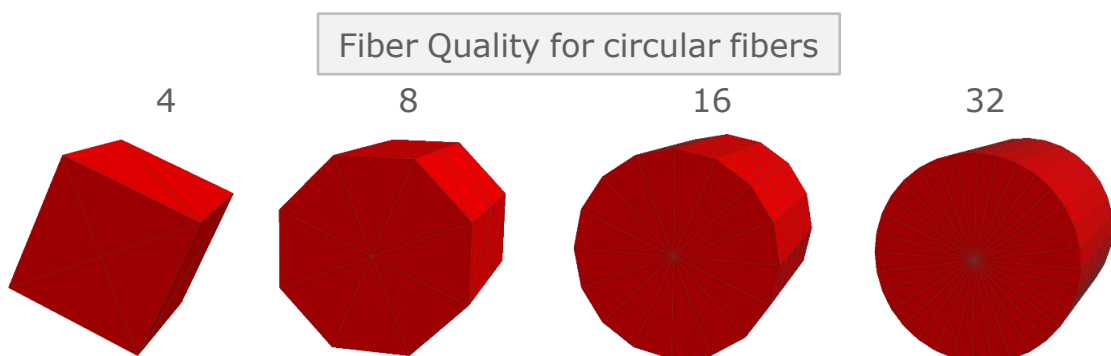
With **Quality Mode** set to **Fixed**, the **Fiber Quality** and **Ellipsoid Quality** are defined globally for the structure. When choosing **Size Dependent**, minimal and maximal values must be defined. This option is recommended if the size of the objects in the structure varies by a large amount.

The quality can only be selected for fibers and ellipsoids. 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).

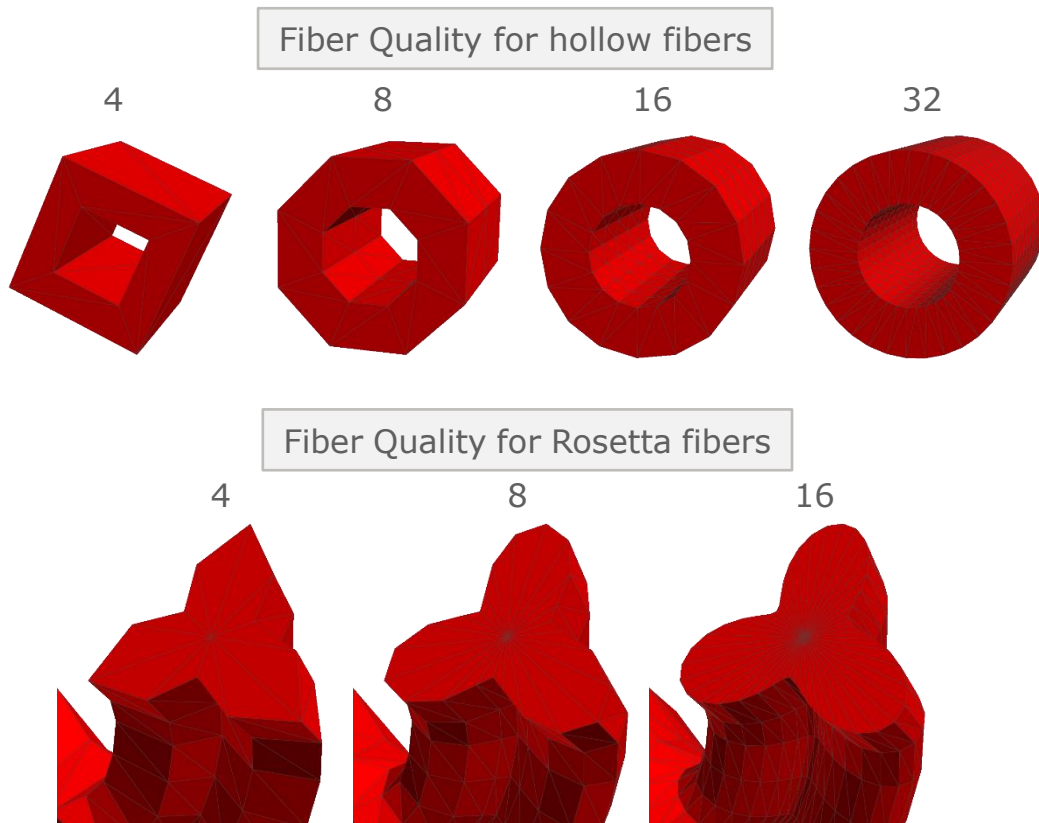
## Fiber Quality

**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.



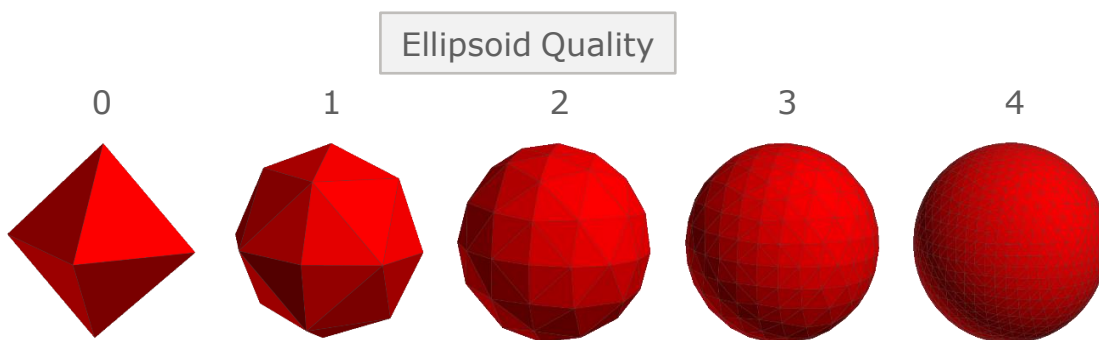




---

### 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  $32 = 8 \times 4$  triangles. For **Ellipsoid Quality = 2**, it contains already  $256 = 8 \times 4^2$  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.


---

### ADD TO CURRENT MESH

See [Add to current mesh](#), in page [10](#).

## PARTICLES MESH

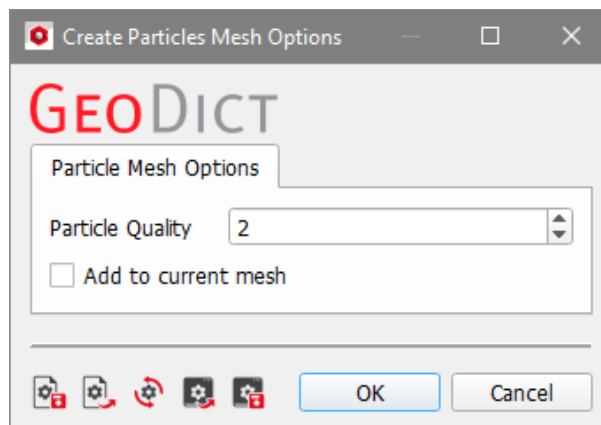
Particle positions and trajectories can be obtained from **FilterDict** and **AddiDict** simulations. **MeshGeo** can create a mesh from the currently loaded particles in **GeoDict**. The mesh is always created at the particle positions currently shown in **GeoDict**. Particles and trajectories can be loaded e.g. from a **FilterDict** or **AddiDict** result file.

To do so, select **File** → **Open Results (\*.gdr)...** from the menu bar or click the  icon in the toolbar. Select a **GeoDict** results file to open it in the Result Viewer. Click **Load Structure** to load the structure for which the results were computed. Under the **Result Visualization** tab, click **Load Particles**.

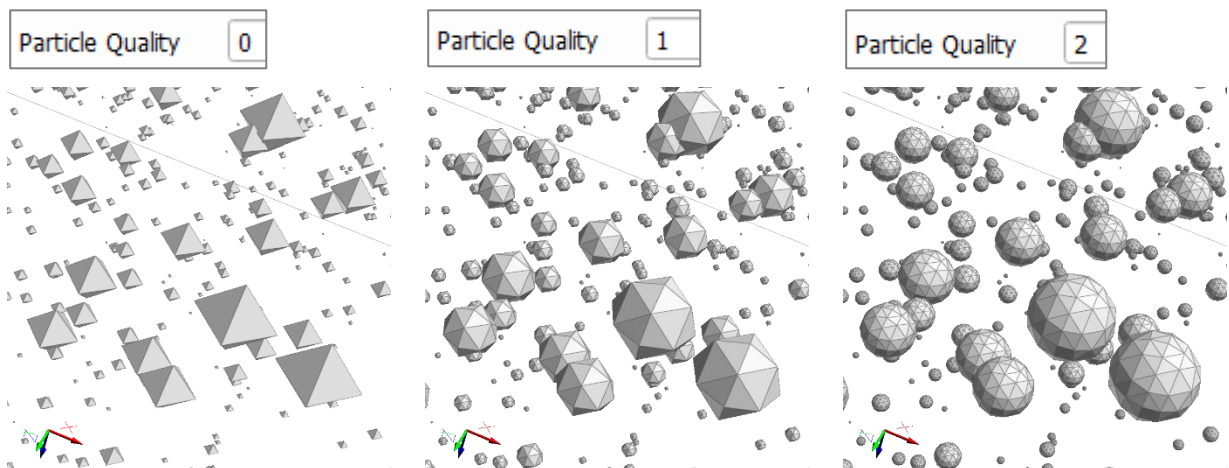
Start **MeshGeo**, select **Create Mesh** and **Particles Mesh** in the **MeshGeo** section and click the **Options' Edit...** button to set the parameters. When all settings are chosen, click the **Run** button at the bottom of the **MeshGeo** section to create the mesh.

## PARTICLE QUALITY

The **Particle Quality** defines the number of triangles in the mesh for each particle. Particles in **GeoDict** are spherical, and the definition of the Particle Quality is the same as the Ellipsoid Quality (see page [14](#)).




Observe the effect of the **Particle Quality** here:



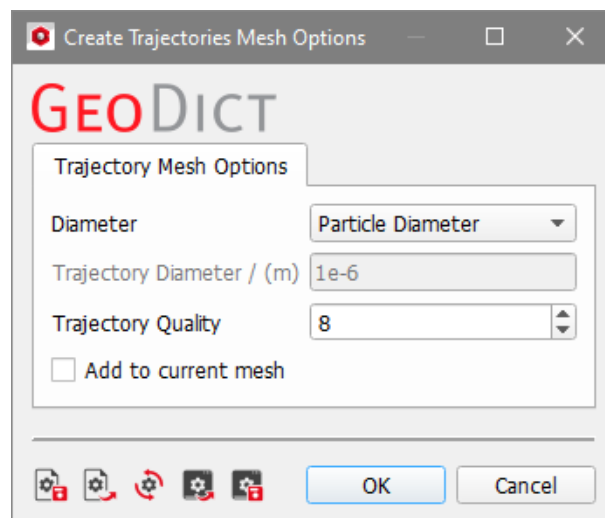
### TRAJECTORIES MESH

As mentioned above, particle positions and trajectories can be obtained from **FilterDict** and **AddiDict** simulations. **MeshGeo** can create a mesh from the currently shown trajectories in **GeoDict**. Particles and trajectories can be loaded e.g. from a **FilterDict** or **AddiDict** result file.

To do so, select **File** → **Open Results (\*.gdr)...** from the menu bar or click the  icon in the toolbar. Select a **GeoDict** results file to open it in the Result Viewer. Click **Load Structure** to load the structure for which the results were computed.

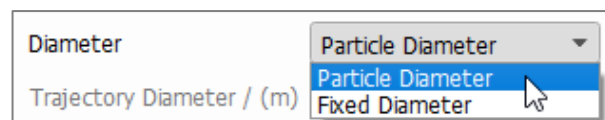
Under the **Particle Visualization (FilterDict)** or **Data Visualization (AddiDict)** tab, make sure that **Trajectories** is checked and click the **Load Particles** button.

Start **MeshGeo**, select **Create Mesh** and **Trajectories Mesh** in the **MeshGeo** section and click the **Edit...** button to set the parameters. When all settings are chosen, click the **Run** button at the bottom of the **MeshGeo** section to create the mesh.



### DIAMETER

The **Diameter** option defines the diameter of the meshed trajectories. The available options are **Particle Diameter** and **Fixed Diameter**. When **Particle Diameter** is chosen, the diameter of the corresponding particles is used. With **Fixed Diameter**, the given **Trajectory Diameter** is used.

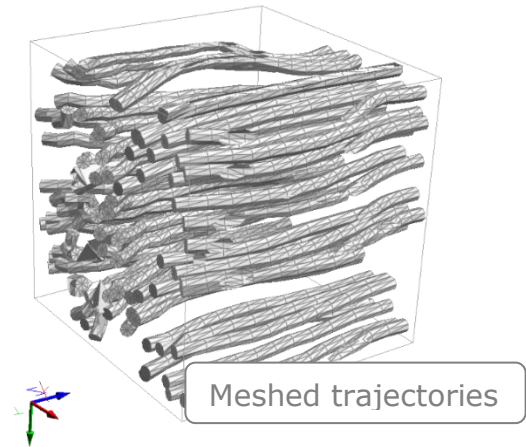
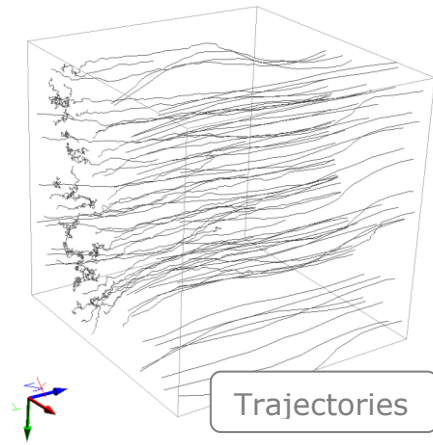


### TRAJECTORY QUALITY

The **Trajectory Quality** sets the quality of the trajectory mesh. It is defined analogously to the **Fiber Quality** for analytic meshes. See page [13](#) for further information.

### ADD TO CURRENT MESH

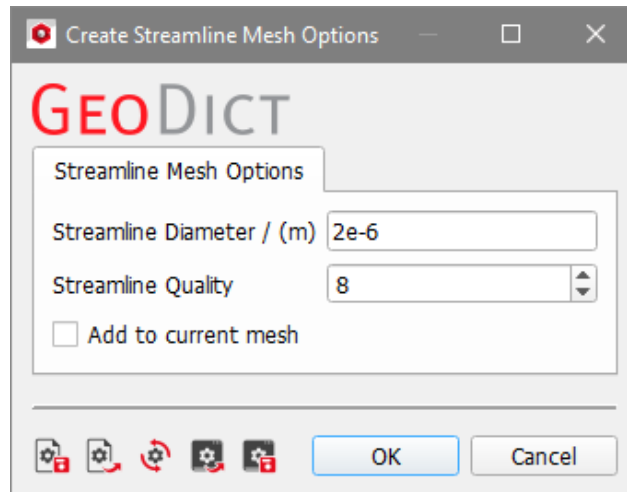
See [Add to current mesh](#) in page [10](#).



## STREAMLINES MESH

[GeoDict](#) can visualize streamlines in flow simulation results (e.g. from [FlowDict](#), [FilterDict](#), [AddiDict](#), [SatuDict](#), and [AcoustoDict](#)). [MeshGeo](#) can create a mesh from the currently shown streamlines in [GeoDict](#).

Start [MeshGeo](#), select **Create Mesh** and **Streamlines Mesh** in the [MeshGeo](#) section and click the **Edit...** button to set the parameters. When all settings are made, click the **Run** button at the bottom of the [MeshGeo](#) section to create the mesh.



### STREAMLINE DIAMETER

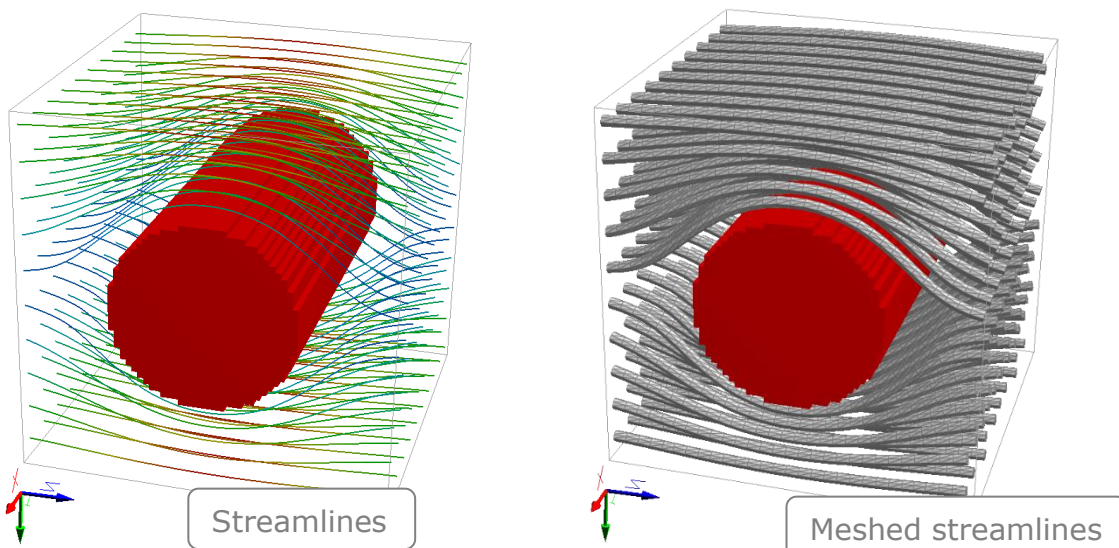
The **Streamline Diameter** option defines the diameter of the meshed streamlines.

### STREAMLINE QUALITY

The **Streamline Quality** sets the quality of the streamlines mesh. It is defined analogously to the [Fiber Quality](#) for round fibers (In **Create Mesh** → **Analytic Mesh**). See page [13](#) for further information.

### ADD TO CURRENT MESH

See [Add to current mesh](#) in page [10](#).



## SAVE MESH

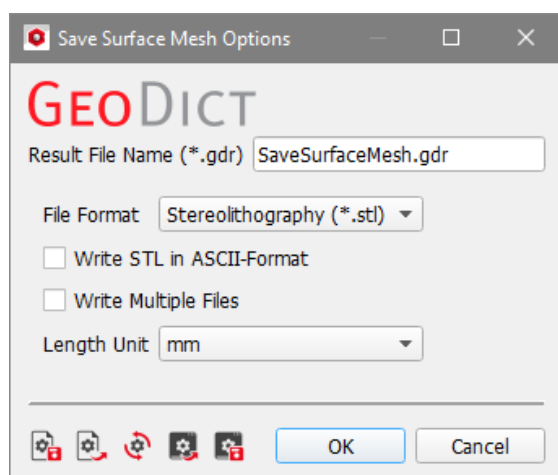
**Save Mesh** writes the currently loaded mesh to a file. When a surface triangulation is loaded, select **Save Mesh**, and then click **Save**.

A dialog opens which allows to select the settings for the saved file. This dialog contains the same options as the **Output Options** tab in **ExportGeo-CAD** (see screenshots below).

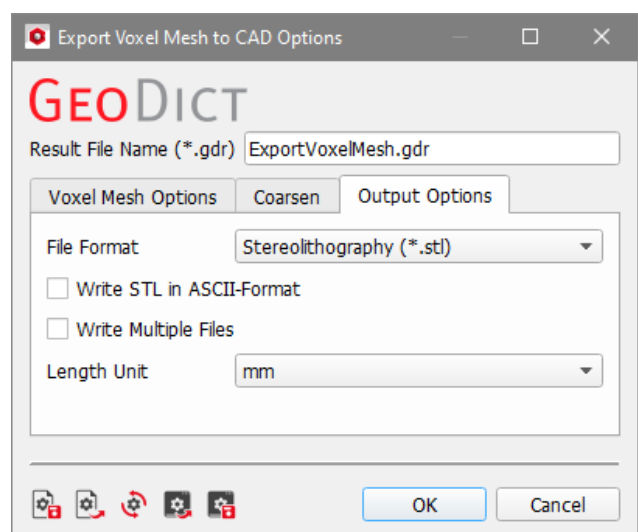
**Save Mesh** can also be reached via **File** → **Save Triangulation as....**



**Save Mesh** options in **MeshGeo**



**Output Options** tab in **ExportGeo-CAD**



## RESULT FILE NAME

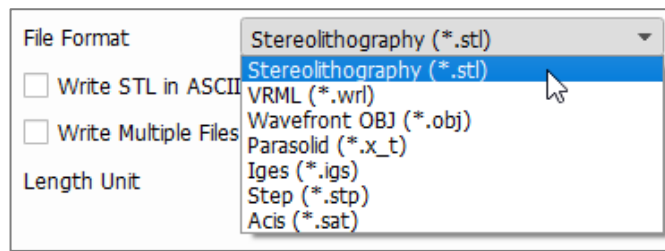
When saving a mesh, a result file is saved in the project folder and contains information about the selected parameters. Select the **Result File Name (\*.gdr)** in the text box. The mesh files are also saved in the project folder.

## FILE FORMAT

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

The [Stereolithography](#) (\*.stl), [VRML](#) (\*.wrl) and [Wavefront OBJ](#) (\*.obj) formats can be exported natively by **GeoDict**. Parasolid (\*.x\_t), Iges (\*.igs), Step (\*.stp) and Acis (\*.sat) require the purchase and installation of the additional software *CADLook*.

If necessary, contact Math2Market for further information.



### ADDITIONAL OPTIONS FOR THE \*.STL FORMAT

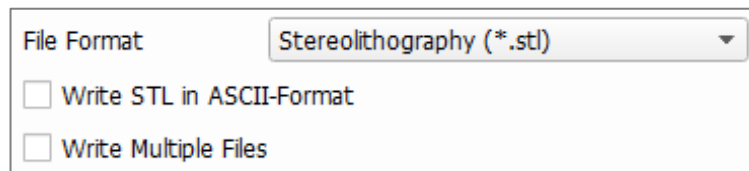
---

If the STL format is chosen, two additional options appear: **Write STL in ASCII format** and **Write Multiple Files**.

#### Write STL in ASCII-Format

---

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

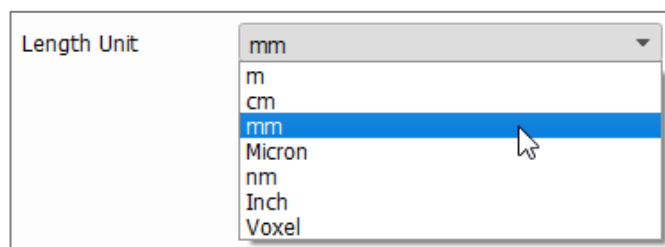
---

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.



```

solid 0
Length Unit m
facet normal 5.836138e-001 7.947828e-001 1.664790e-001
  outer loop
    vertex 2.622470e-005 8.443399e-005 4.803175e-005
    vertex 3.167581e-005 8.211901e-005 3.997407e-005
    vertex 3.098098e-005 8.342811e-005 3.616017e-005
  endloop
endfacet
facet normal 5.836138e-001 7.947828e-001 1.664790e-001
  outer loop
    vertex 2.622470e-005 8.443399e-005 4.803175e-005

```

```

solid 0
Length Unit mm
facet normal 5.836138e-001 7.947828e-001 1.664790e-001
  outer loop
    vertex 2.622470e-002 8.443399e-002 4.803175e-002
    vertex 3.167581e-002 8.211901e-002 3.997407e-002
    vertex 3.098098e-002 8.342811e-002 3.616017e-002
  endloop
endfacet
facet normal 5.836138e-001 7.947828e-001 1.664790e-001
  outer loop
    vertex 2.622470e-002 8.443399e-002 4.803175e-002

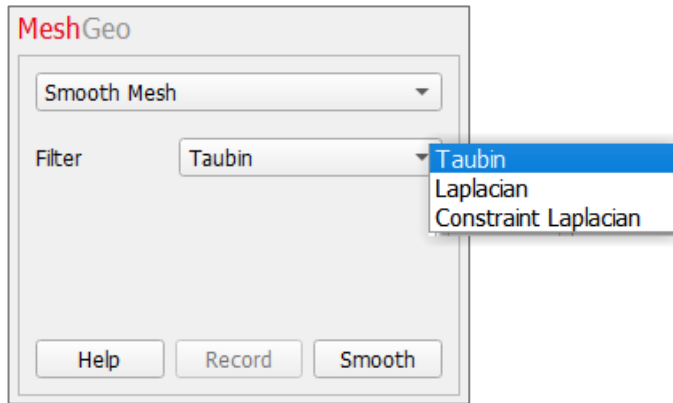
```



## SMOOTH MESH

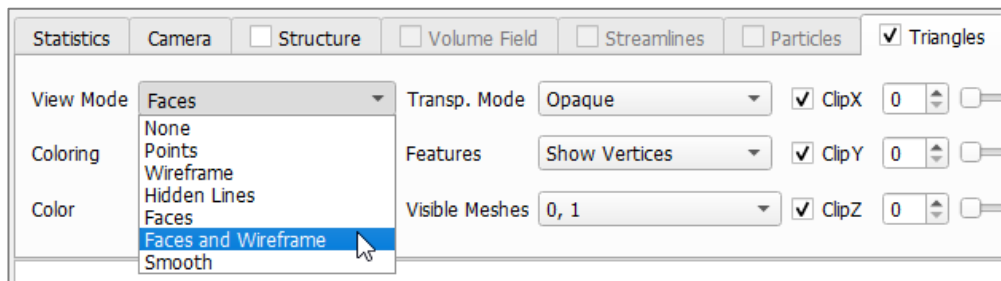
Meshes from voxel structures often have a significant surface roughness which can be reduced by applying smoothing filters.

In MeshGeo, three smoothing filters are selectable from the pull-down menu to improve the mesh quality: **Taubin**, **Laplacian**, or **Constraint Laplacian**. Choose a filter and click **Smooth** to apply the selected filter to the current mesh.

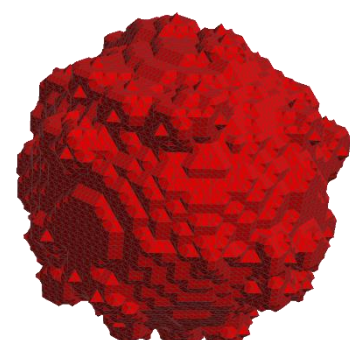
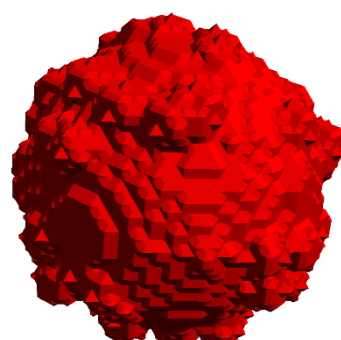
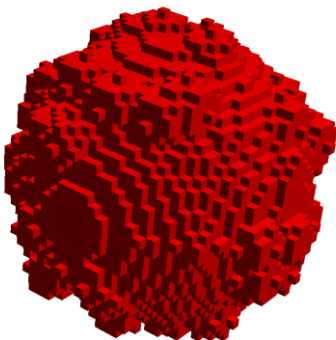
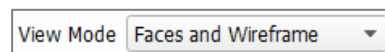
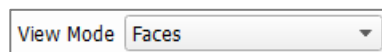


Smoothing is best shown on a structure with an irregular surface. The example structure created here is a grain with surface roughness. The original mesh before smoothing is created with **Create Mesh** → **Voxel Mesh** and the [Multi Material](#) mode (see page 9).

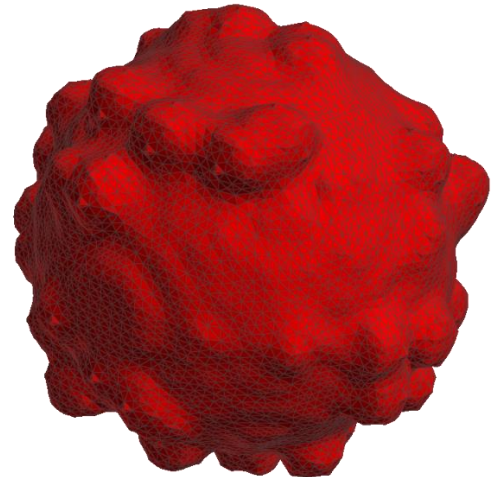
To observe the difference between the different mesh types, it is recommended to set the **View Mode** to **Faces and Wireframe** (**Triangles** tab → **View Mode** → **Faces and Wireframe**, in the Visualization panel above the Visualization area).



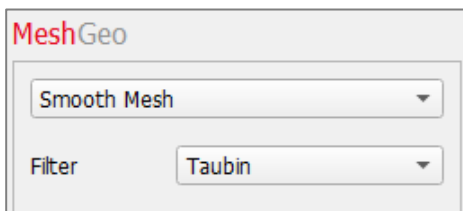
Voxel structure



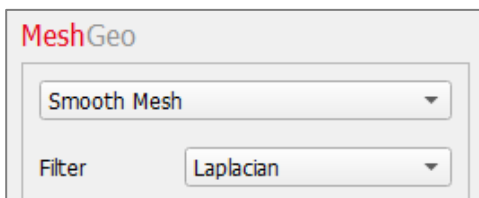
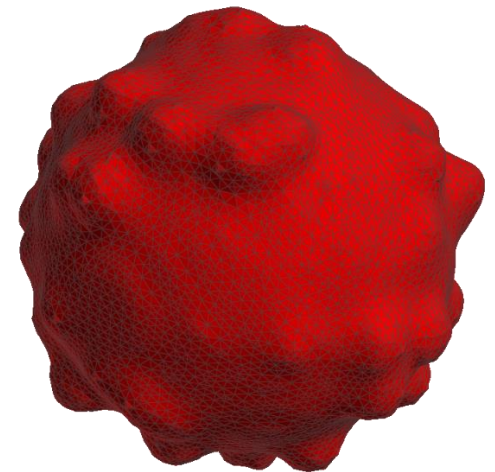
The **Taubin** filter is the default smoothing filter in GeoDict. This filter iteratively combines different Laplacian filters to provide a better volume preservation. The runtime for **Taubin** is longer than for Laplacian. In the screenshot on the right, observe that the mesh is rougher than the result after **Laplacian** filtering, while its volume is closer to the volume of the original voxel structure.



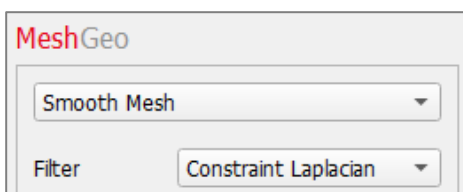
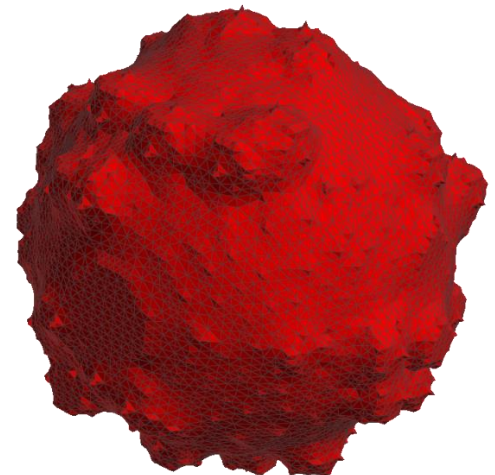
For further information about the Taubin filter, see [Reference (1)].



The **Laplacian** filter is a simple and fast option for mesh smoothing. However, this filter causes mesh shrinking for most geometries. For further information, see [Wikipedia: Laplacian Smoothing](#).



The **Constraint Laplacian** filter is a derivation of the Laplacian approach, but it ensures that points from the triangulation stay in the voxel they originated from. This results in an even better preservation of the volume, but it can also lead to artefacts in the mesh (Observe the small peaks on the surface in the example on the right).



## References

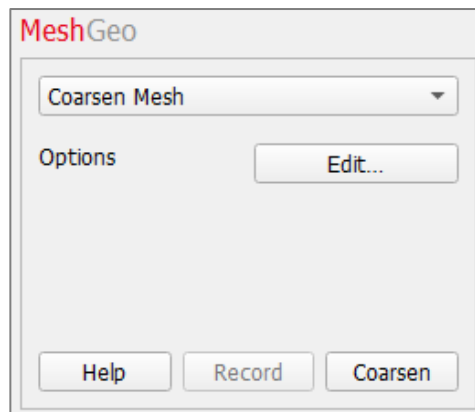
(1) Curve and surface smoothing without shrinkage. *Proceedings of IEEE international conference on computer vision* (pp. 852-857). [doi:10.1109/ICCV.1995.466848](https://doi.org/10.1109/ICCV.1995.466848)

## COARSEN MESH

**Coarsen Mesh** in MeshGeo reduces the number of triangles in the currently loaded surface mesh.

When a surface triangulation is loaded, select **Coarsen Mesh** and click **Edit** to open the **Coarsen** dialog.

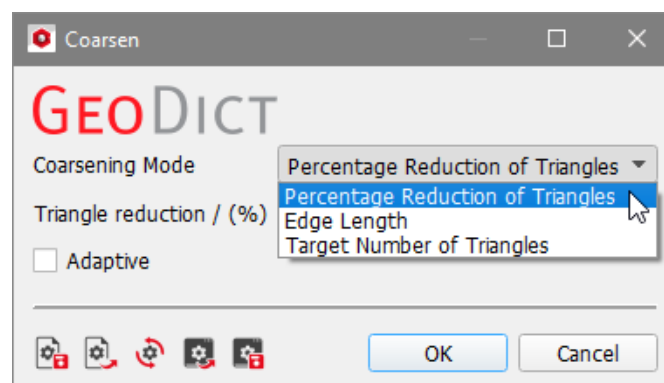
Generally, it is not supported to use Coarsening on the voxel surface mesh (**Create Mesh** → **Voxel Mesh** → **Voxel Surface**). Use the other voxel mesh modes or previous mesh smoothing instead.



## 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**.

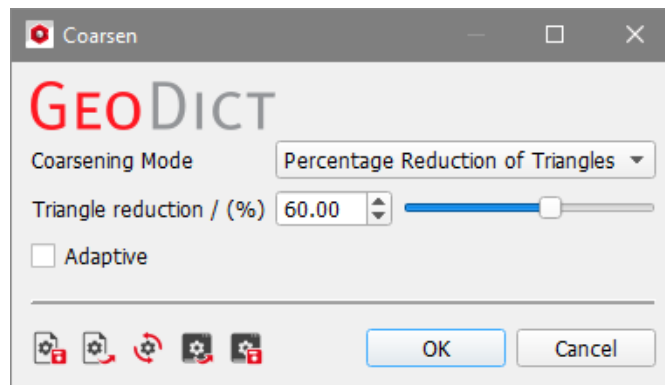
The coarsening algorithm reduces the number of triangles in the currently loaded mesh by combining short edges to longer edges.



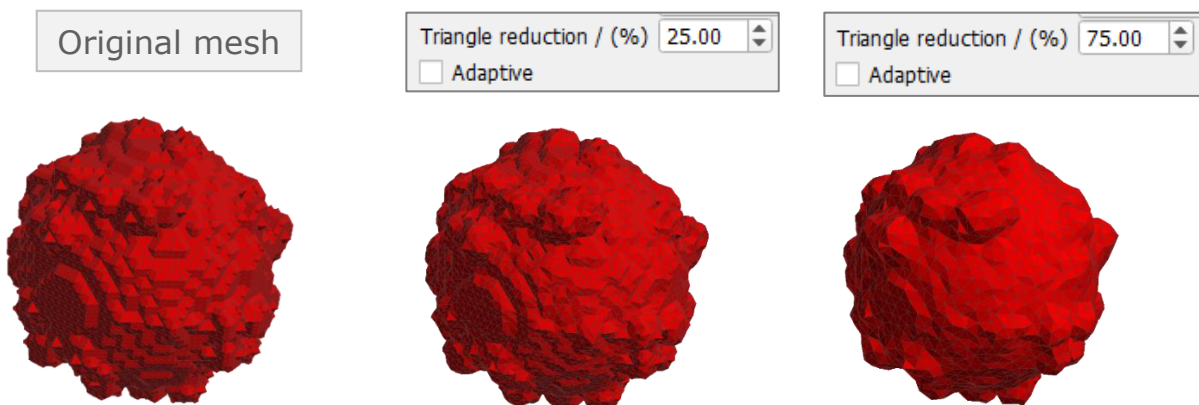
### PERCENTAGE REDUCTION OF TRIANGLES

**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, use the option **Triangle reduction / (%)**.



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

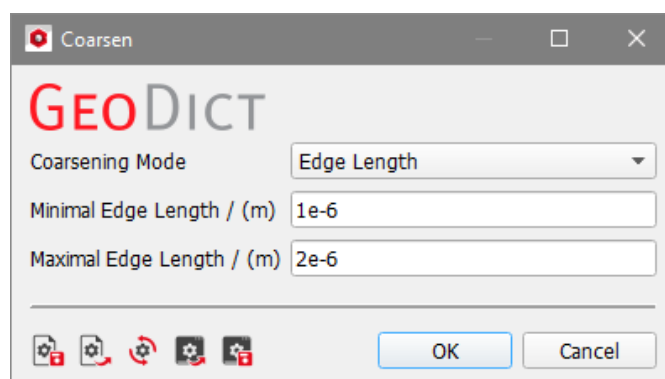


### EDGE LENGTH

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

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



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

*Nevertheless, it is not always possible to satisfy the given maximal edge length.*

#### TARGET NUMBER OF TRIANGLES

---

**Target Number of Triangle** 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.

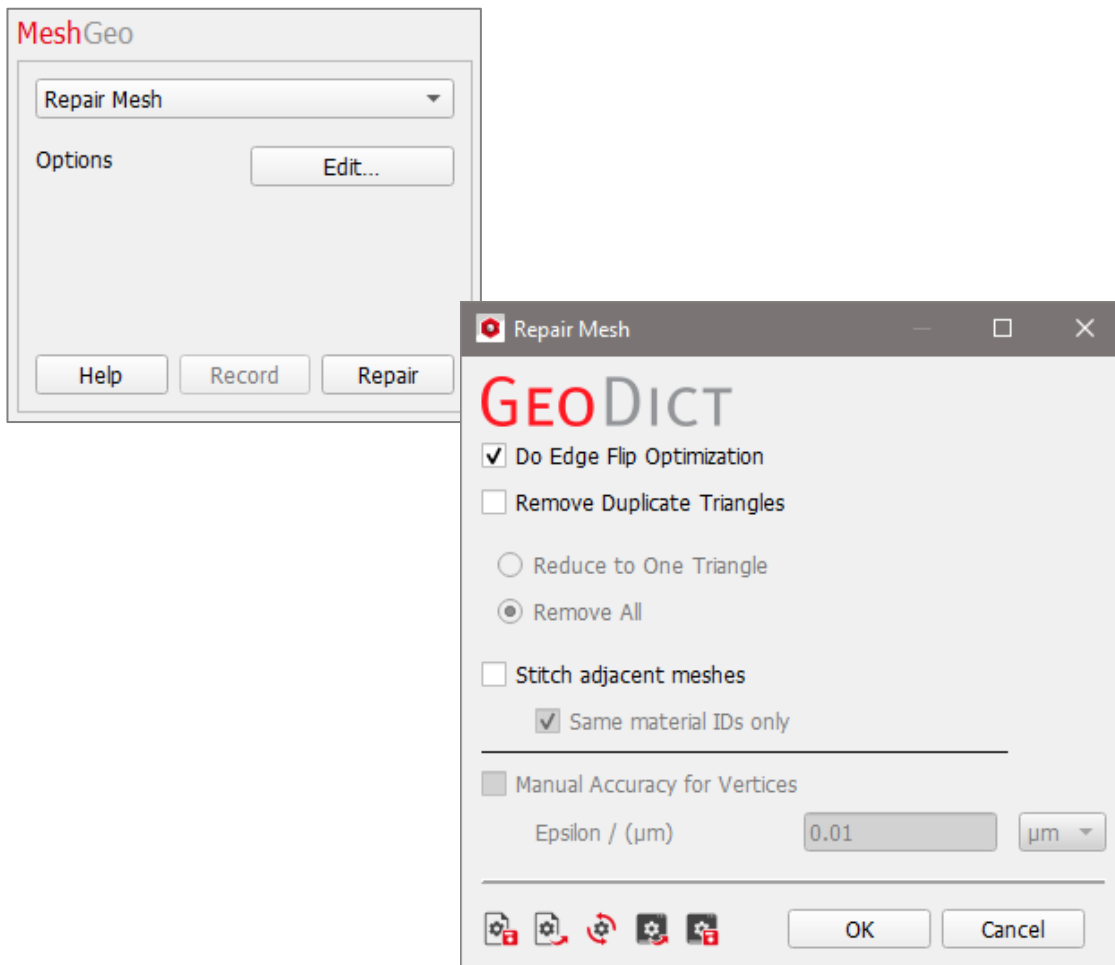
When the **Adaptive** mode is disabled, the coarsening algorithm works based on the edge lengths in the mesh.

## REPAIR MESH

**Repair Mesh** provides options to repair meshes. The parameters for repairing the mesh can be defined by clicking the **Edit...** button.

Repair Mesh is only available in **MeshGeo**, since it works on the mesh currently loaded in GeoDict. With **ExportGeo-CAD**, the meshes are saved directly and not loaded in the GeoDict user interface.

When a surface triangulation is loaded, select **Repair Mesh** and click **Edit...** to open the **Repair Mesh** dialog.

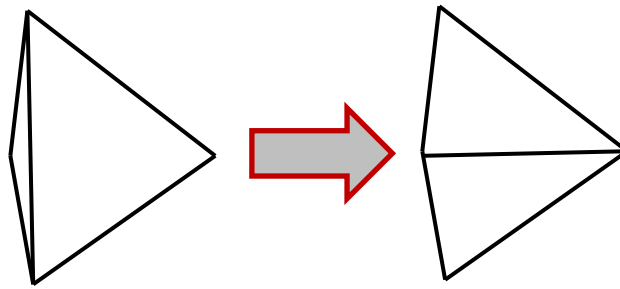


When all settings have been chosen, click **Repair** at the bottom of the **MeshGeo** section to create the mesh.

## DO EDGE FLIP OPTIMIZATION

With **Do Edge Flip Optimization**, folded edges in the mesh are analyzed and repaired by edge flipping. Folded edges are edges which meet in an angle close to  $0^\circ$ , which can cause problems when further working with the mesh.

Edge flipping refers to switching the end points of an edge in two adjacent triangles, which can help to avoid small angles. A simple example is shown in the screenshot below, where one of the triangles on the left has a very small angle. By flipping the edges, this problem is resolved.



## REMOVE DUPLICATED TRIANGLES

---

The duplicated triangles are checked and removed with **Remove Duplicate Triangles**. Two possibilities are possible for the removing. One is **Reduce to One Triangle**, which leaves one instance of each set of duplicate triangles. The other one is **Remove All**, which removes all instances of each set of duplicate triangles.

## STITCH ADJACENT MESHES

---

With **Stitch adjacent meshes**, the meshes with overlapping boundary edges are merged. If **Same material IDs** only is checked, the merge is done only when the material IDs of the overlapping meshes are the same. Otherwise, the meshes with different material IDs are merged, too.

## MANUAL ACCURACY FOR VERTICES

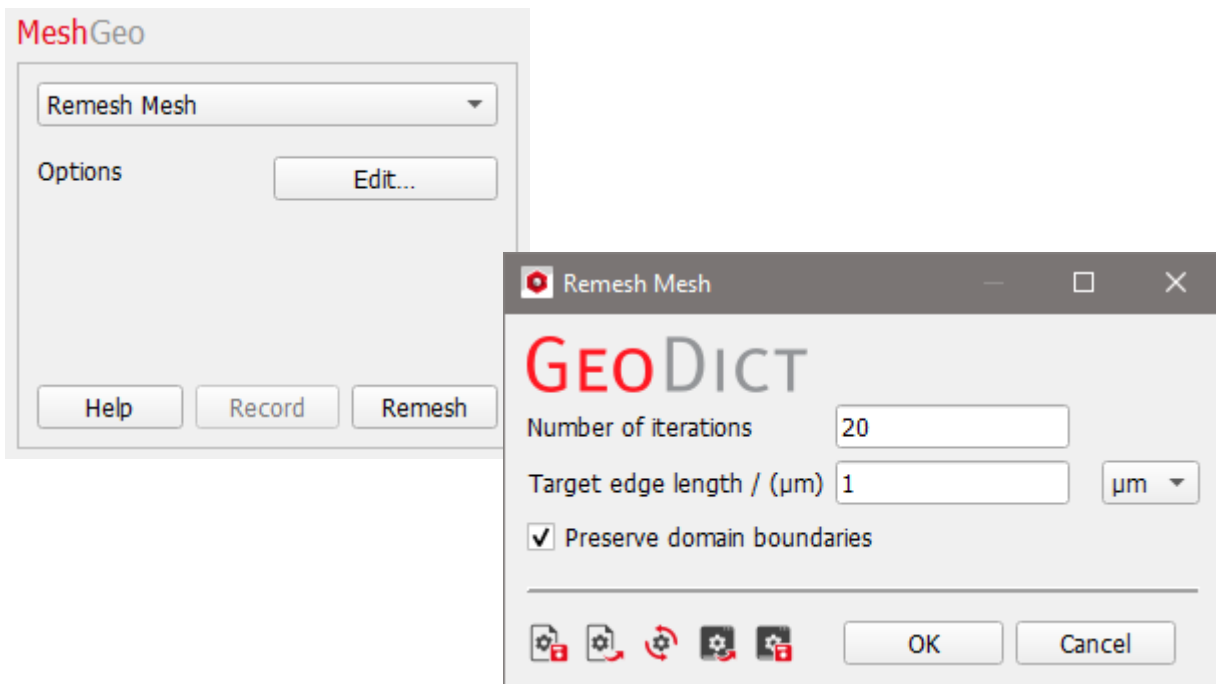
---

The option of **Manual Accuracy for Vertices** is valid only when **Remove Duplicate Triangles** is used to define a threshold to compensate for numerical inaccuracies in vertex coordinates. The mesh vertices with at most the distance given for **Epsilon / ( $\mu\text{m}$ )** is treated as equal.



## REMESH MESH

The loaded mesh can be re-meshed with given edge length of triangles. click **Edit** to open the **Remesh Mesh** dialog.



### NUMBER OF ITERATIONS

The **Number of Iterations** option defines the number of iterative meshing steps. In each iteration, the mesh is proceeded such that edge lengths are converging into the given target value and vertices are evenly distributed on the surface.

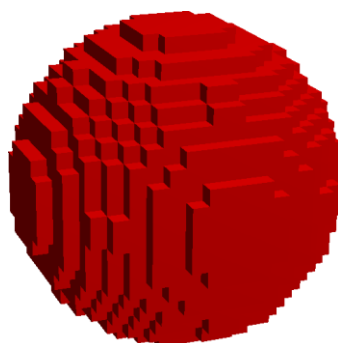
### TARGE EDGE LENGTH

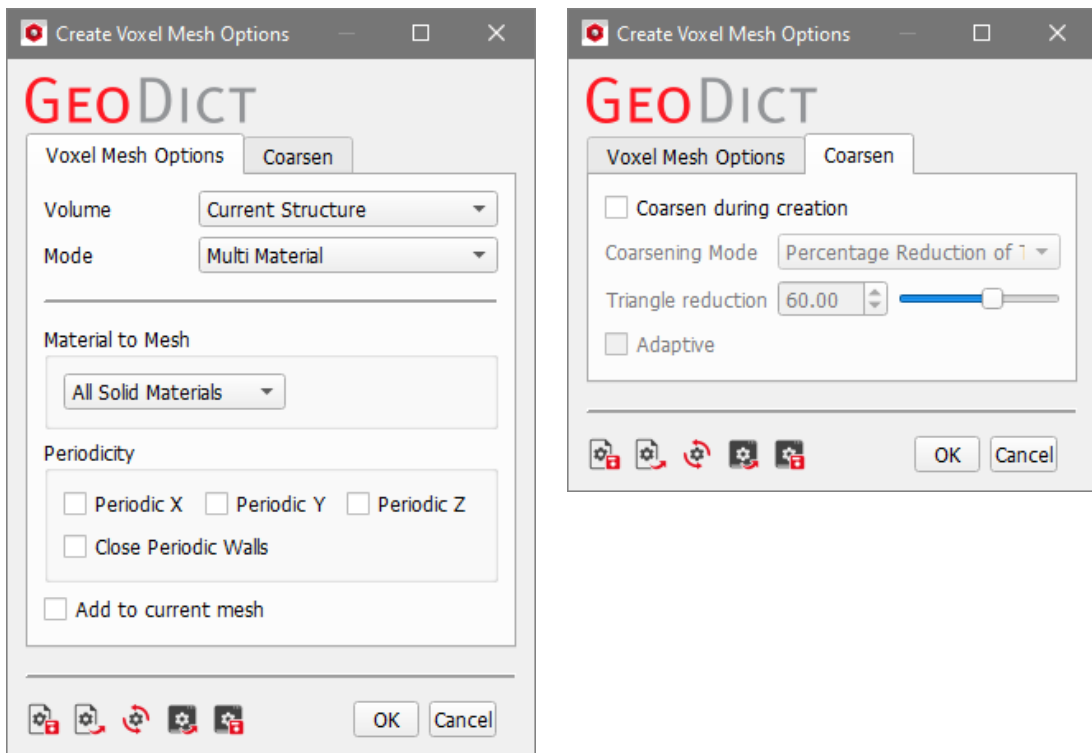
The remeshing aims to optimize the mesh such that the edge length converges to the **Target edge length**.

### PRESERVE DOMAIN BOUNDARIES

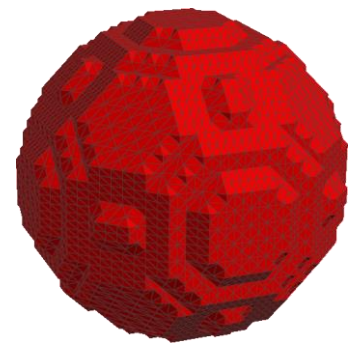
If **Preserve domain boundaries** is checked, the mesh vertices that are located on the domain's boundary will remain on the boundary after remeshing.

To illustrate the features, an example of a ball with 20 µm diameter with the voxel length 1 µm is used for meshing (**Create Mesh**), remeshing (**Remesh Mesh**), and smoothing (**Smooth Mesh**) to show the different meshes created.

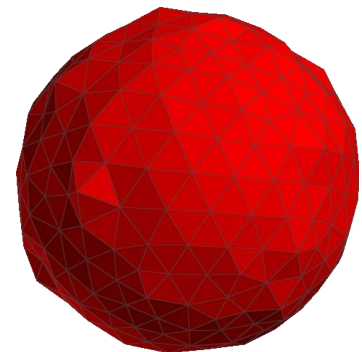
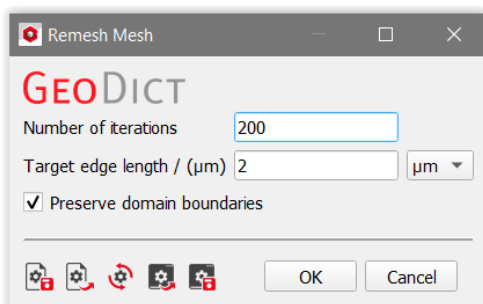




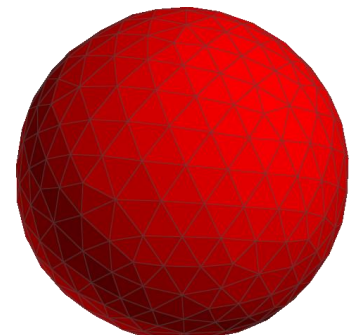
First, a voxel mesh is created with the settings shown in the dialogs above, without any **Coarsen**.



**Remesh Mesh** is executed with 200 as the **Number of iterations** and 2  $\mu\text{m}$  as the **Target edge length**.



The mesh is then smoothed with the **Taubin** filter.

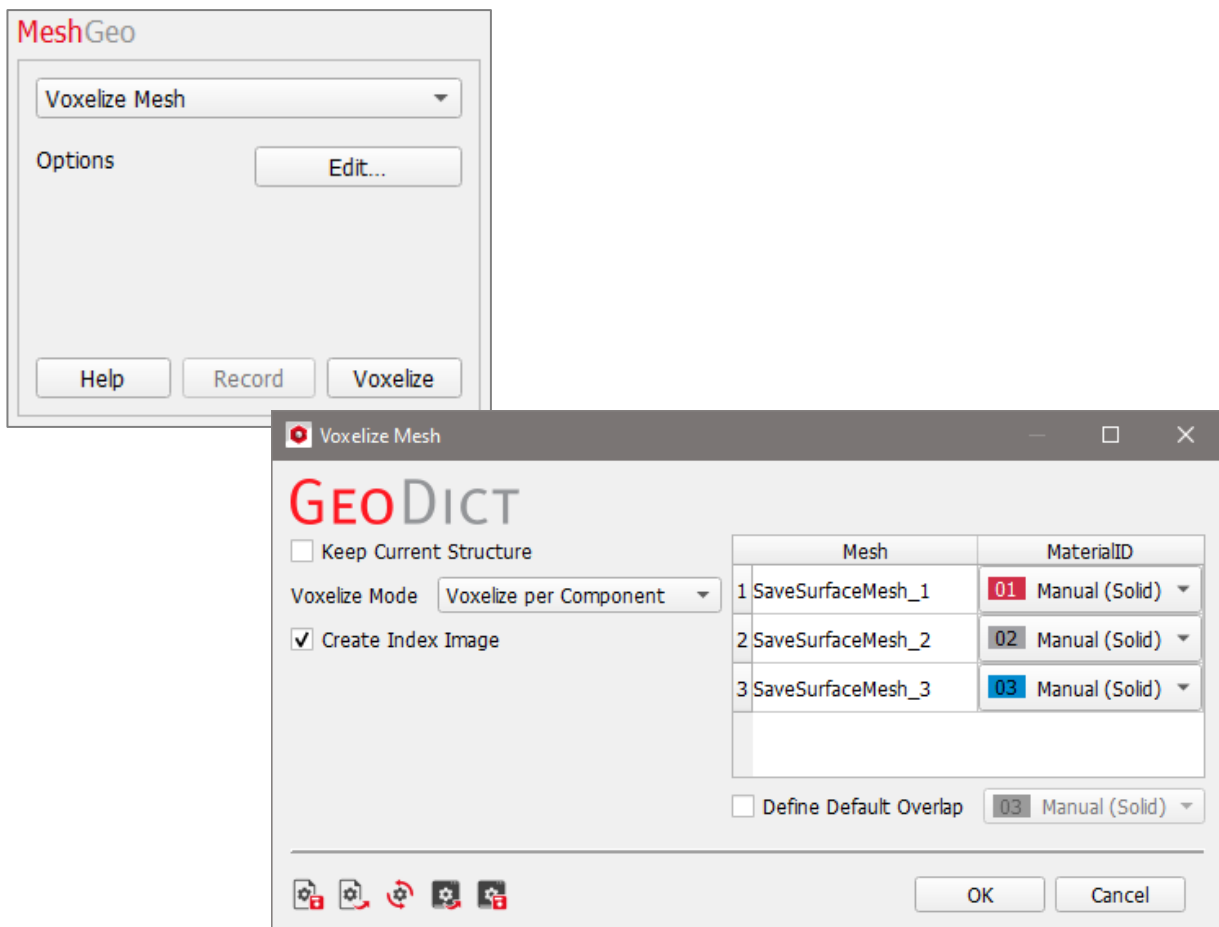


## VOXELIZE MESH

With **Voxelize Mesh**, a voxel structure is generated based on the currently loaded meshes. **Voxelize Mesh** provides the same functionality as **ImportGeo-CAD**, the only significant difference is that **ImportGeo-CAD** works on the meshes during their import, while **Voxelize Mesh** works on the meshes which are already in memory.

**Voxelize Mesh** is only available in **MeshGeo**, since it works on the mesh currently loaded in **GeoDict**. With **ExportGeo-CAD**, the meshes are saved directly and not loaded in the **GeoDict** user interface.

When a surface triangulation is loaded, select **Voxelize Mesh** and click **Edit...** to open the **Voxelize Mesh** dialog. When all settings have been chosen, click **Voxelize** at the bottom of the **MeshGeo** section to create the mesh.



The settings in the **Voxelize Mesh** dialog are analogous to the **ImportGeo-CAD** dialog. Please refer to the [ImportGeo-CAD](#) handbook for further explanation of the settings. The settings which correspond to the domain size are not available here since these are taken from the currently loaded structure.

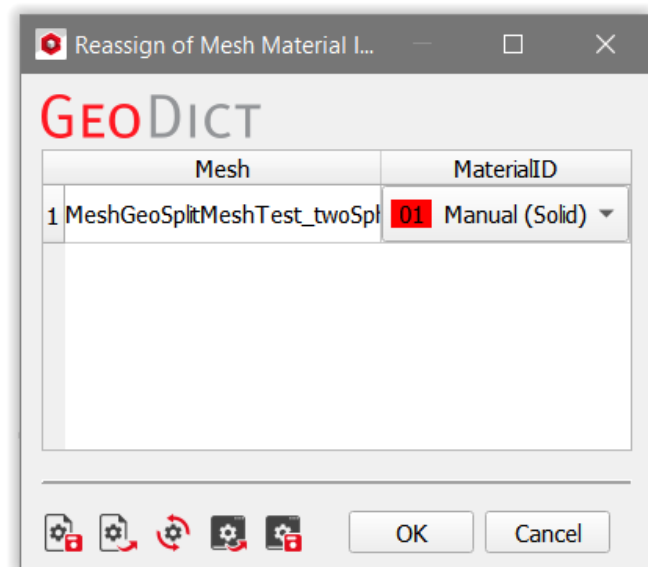
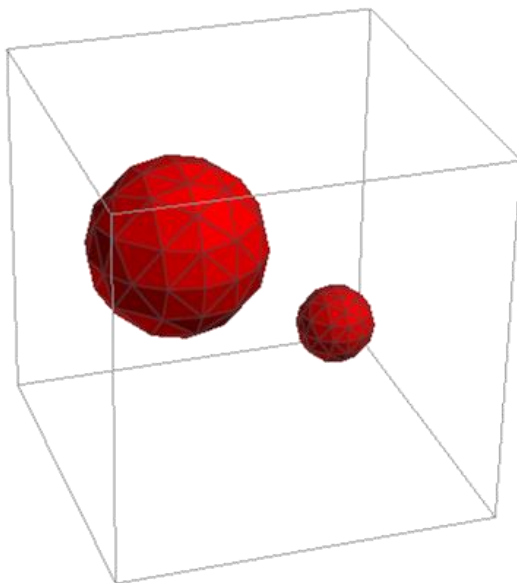
## SPLIT MESHES INTO CONNECTED COMPONENTS

With **Split Meshes into Connected Components**, an individual mesh handle is created for each connected component of the loaded surface triangulation.



For example, a surface mesh loaded from a single .stl file as shown below for two spheres. There is a single mesh handle for all the objects.

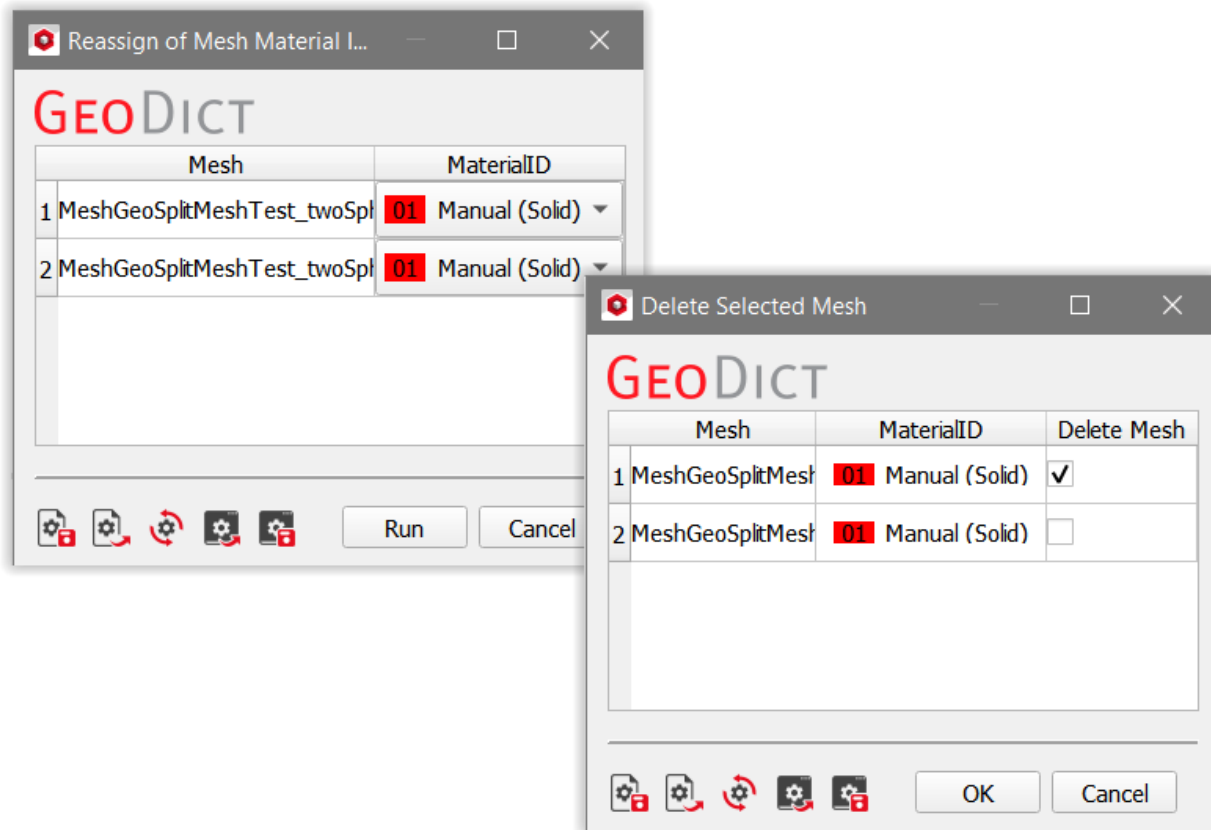
Open the settings dialog for **Reassign Mesh Material ID** (see below in page [36](#)) and observe that only one mesh is shown.



Now come back to **Split Meshes into Connected Components**. Since the command detects the connected components automatically and separates them into individual mesh, no settings are necessary. Here just click **Split** to make it work.

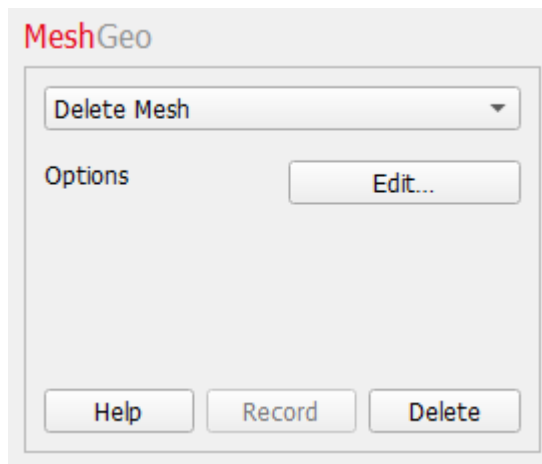
When opening the dialog of **Reassign Mesh Material ID** again, two mesh handles now appear for the two spheres.

Now, separately, the two meshes can be saved (**Save Mesh**), deleted (**Delete Mesh**), or assigned to different Material IDs (**Reassign Mesh Material ID**).

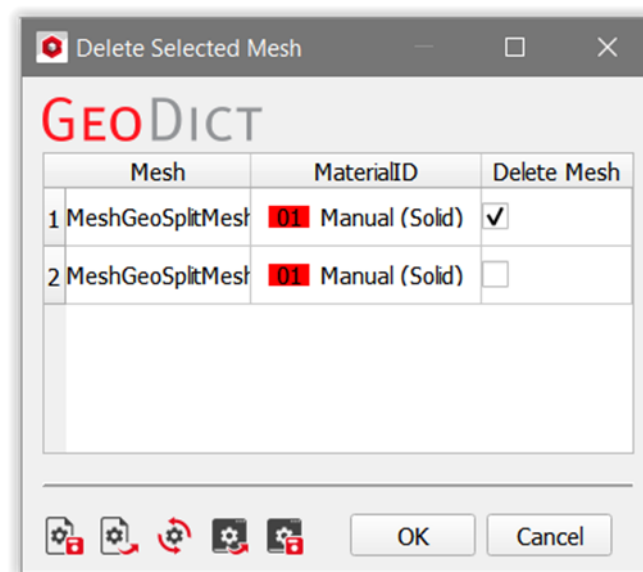


## DELETE MESH

The selected meshes can be deleted with **Delete Mesh**.



Click **Edit...** to open the **Delete Selected Mesh** dialog, the meshes currently loaded in memory are shown in the **Mesh** column.

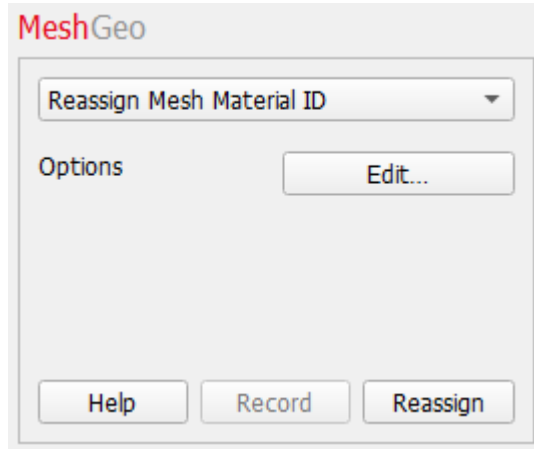


Click the checkboxes under **Delete Mesh** column for the meshes needed to be deleted, then click **OK** to go back to the MeshGeo section.

Click **Delete** to remove the selected meshes.

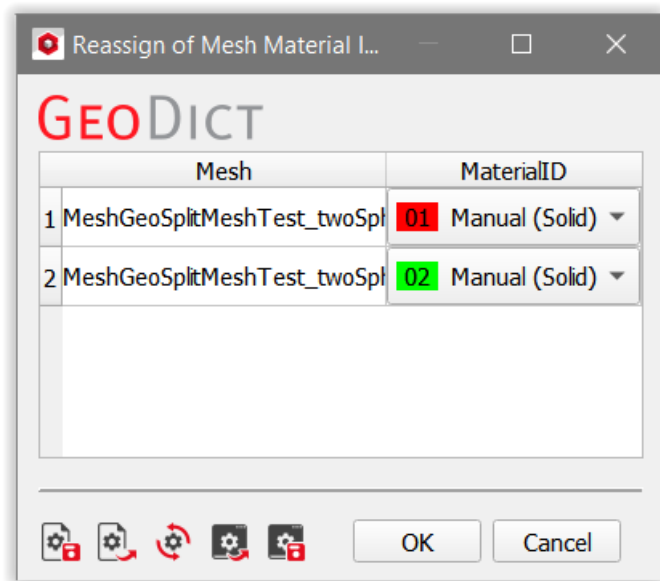
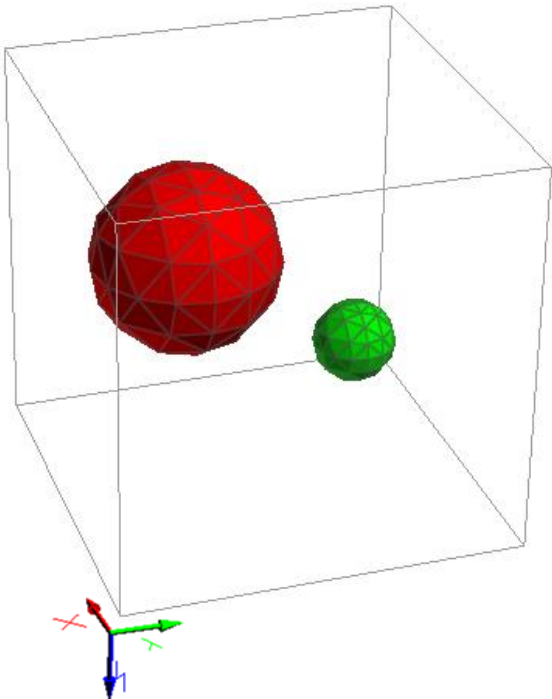
## REASSIGN MESH MATERIAL ID

Each mesh can be assigned to a different material ID with **Reassign Mesh Material ID**.



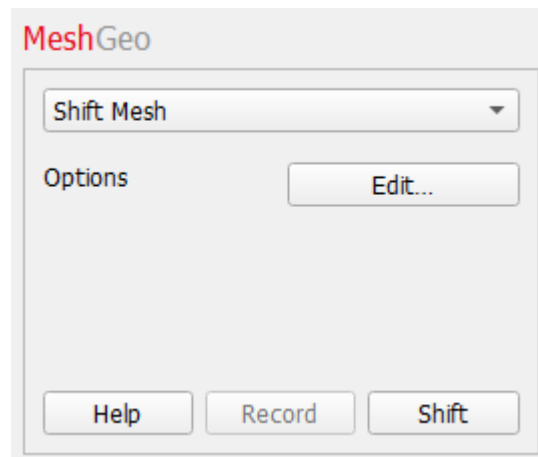
For the example of two spheres shown in **Split Meshes into Connected Components**, (page 33) click **Edit...** to open the dialog.

Choose Material 02 for the small sphere 2, and click **OK**. Click **Reassign** in MeshGeo section. The two spheres now have different material IDs.

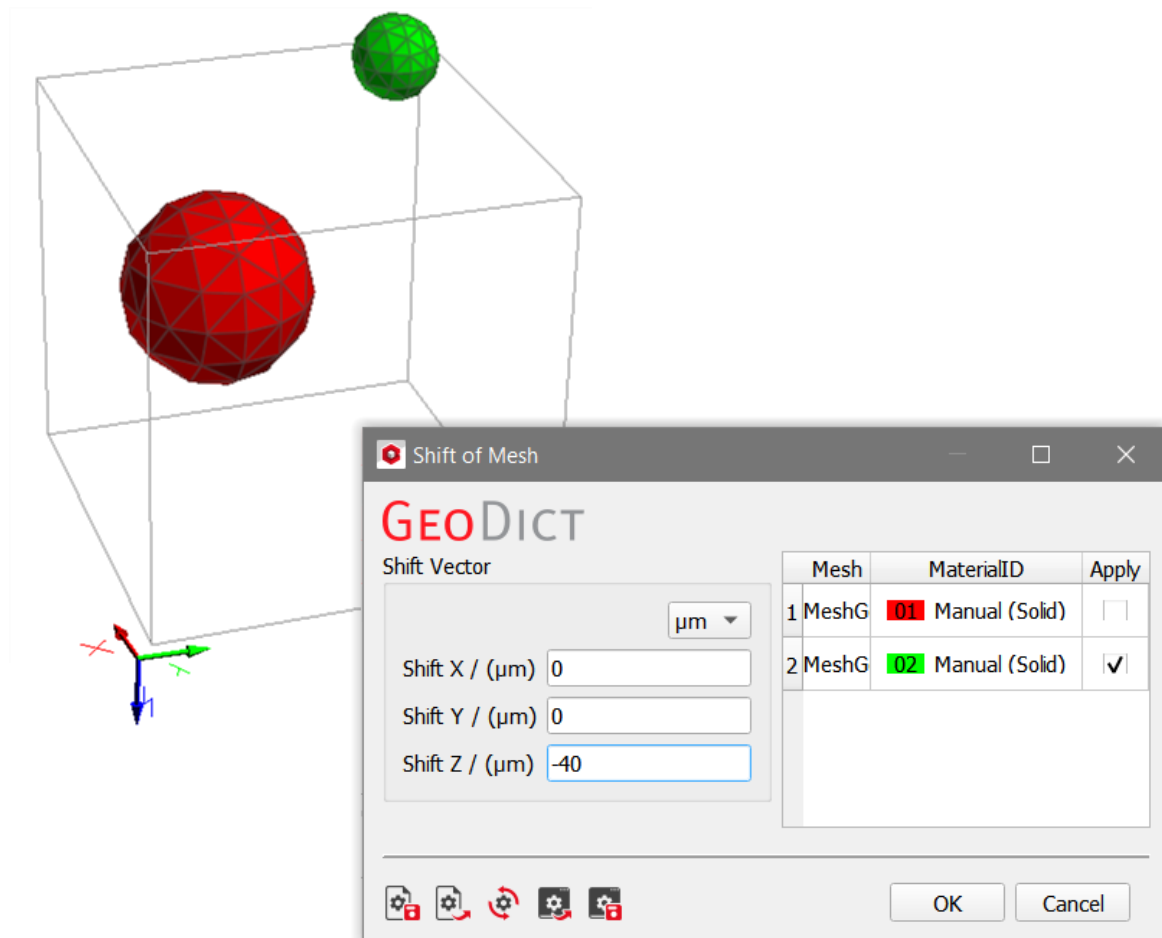


## SHIFT MESH

Meshes can be shifted with a given distance with **Shift Mesh**.



Select **Shift Mesh** and click **Edit...** to open the **Shift Mesh** dialog. Choose the mesh to be shifted on the right box and give the values for **Shift X**, **Shift Y**, and **Shift Z** for the shift distance in X, Y, and Z direction, respectively. Click **OK** to close the dialog and go back to MeshGeo section. Click **Shift** to move the green sphere in the negative Z-direction by a distance of 40 $\mu\text{m}$ .



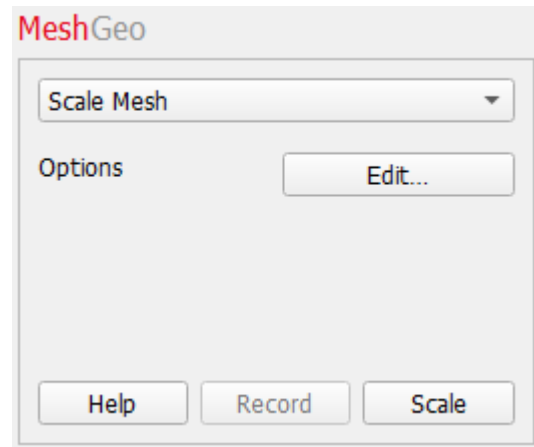


## SCALE MESH

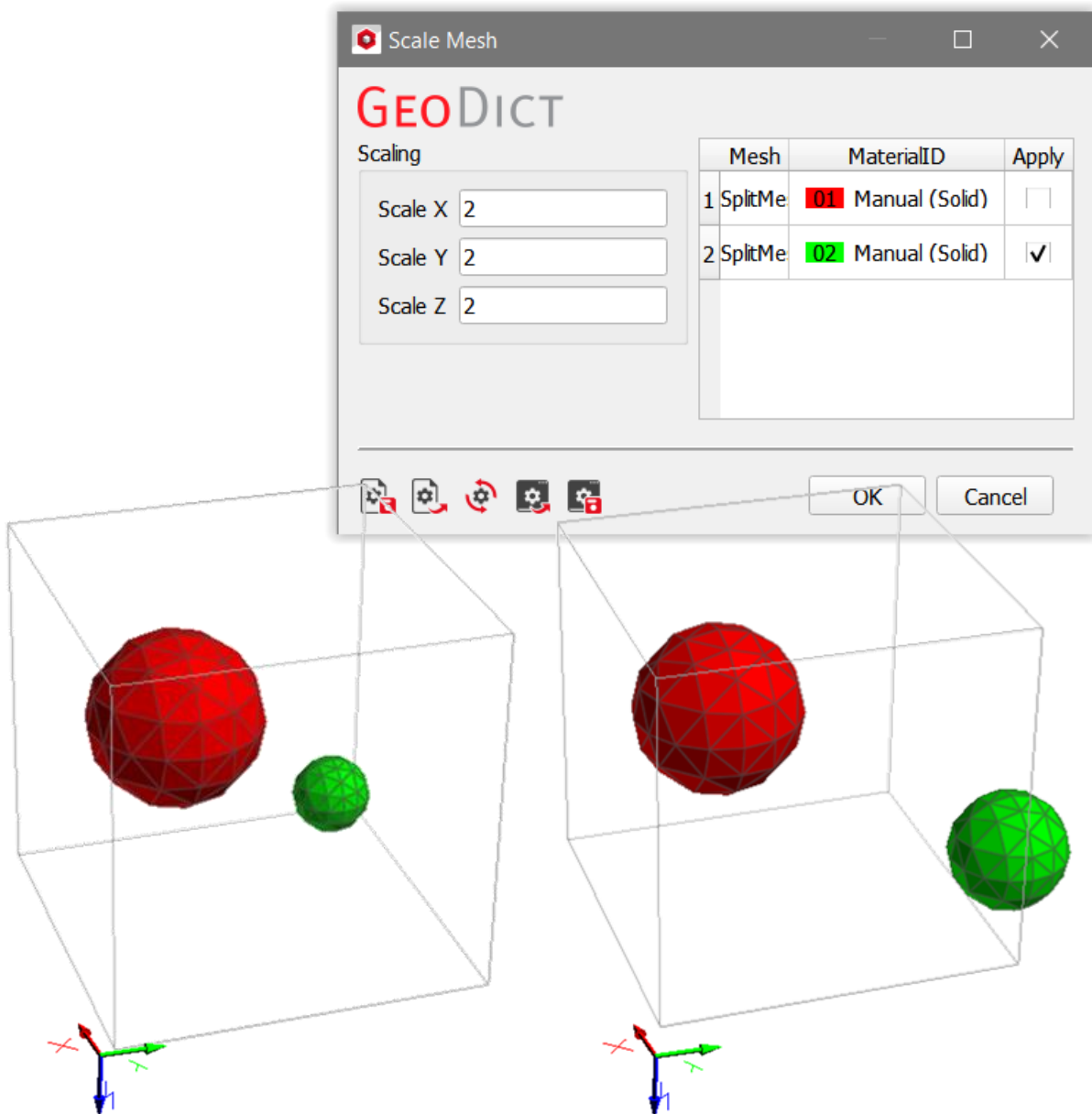
The mesh can be scaled with **Scale Mesh**.

Select **Scale Mesh** and click **Edit** to open the dialog of **Scale Mesh**.

Choose the mesh to be shifted on the right box and give the scaling factors for x, y, and z coordinate in **Scale X**, **Scale Y**, and **Scale Z**.



For the two sphere example from the previous page, the comparison of the meshes before and after a scaling of 2 for x, y, and z are shown below.



If the mesh of the green particle is saved, it can be observed that the coordinates of x, y, and z after scaling are two times of the original values.

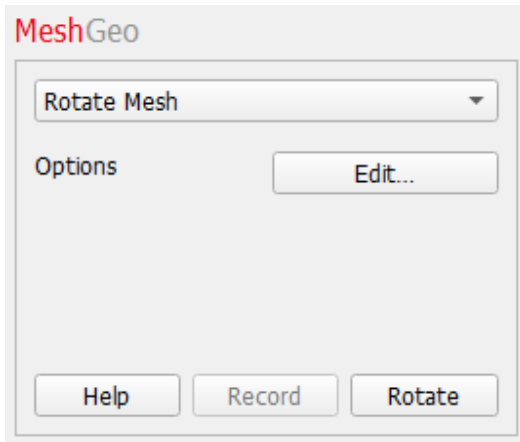
Here, below, the first image shows the first lines of values in STL before scaling. The second image shows the values after scaling.

```
solid 1
  facet normal +9.626382e-01 -1.914781e-01 +1.914781e-01
    outer loop
      vertex 4.461940e-02 4.000000e-02 4.191342e-02
      vertex 4.461940e-02 3.808658e-02 4.000000e-02
      vertex 4.500000e-02 4.000000e-02 4.000000e-02
    endloop
  endfacet
  facet normal +8.177513e-01 -1.808815e-01 +5.464108e-01
    outer loop
      vertex 4.353553e-02 4.000000e-02 4.353553e-02
      vertex 4.408249e-02 3.795876e-02 4.204124e-02
      vertex 4.461940e-02 4.000000e-02 4.191342e-02
    endloop
  endfacet
```

```
solid 1
  facet normal +9.626382e-01 -1.914781e-01 +1.914781e-01
    outer loop
      vertex 8.923881e-02 8.000000e-02 8.382684e-02
      vertex 8.923881e-02 7.617316e-02 8.000000e-02
      vertex 9.000000e-02 8.000000e-02 8.000000e-02
    endloop
  endfacet
  facet normal +8.177513e-01 -1.808815e-01 +5.464108e-01
    outer loop
      vertex 8.707106e-02 8.000000e-02 8.707106e-02
      vertex 8.816497e-02 7.591752e-02 8.408248e-02
      vertex 8.923881e-02 8.000000e-02 8.382684e-02
    endloop
  endfacet
```

## ROTATE MESH

If a mesh needs to be rotated, select **Rotate Mesh** from the pull-down menu and click **Edit...** to open the **Rotate Mesh** dialog.



Enter the [Euler Angles](#) **Phi**, **Theta**, **Psi** by which the selected should rotate. More details about Euler Angles can be found in [GadGeo](#) handbook.

The rotation can be around either the domain center or domain origin.

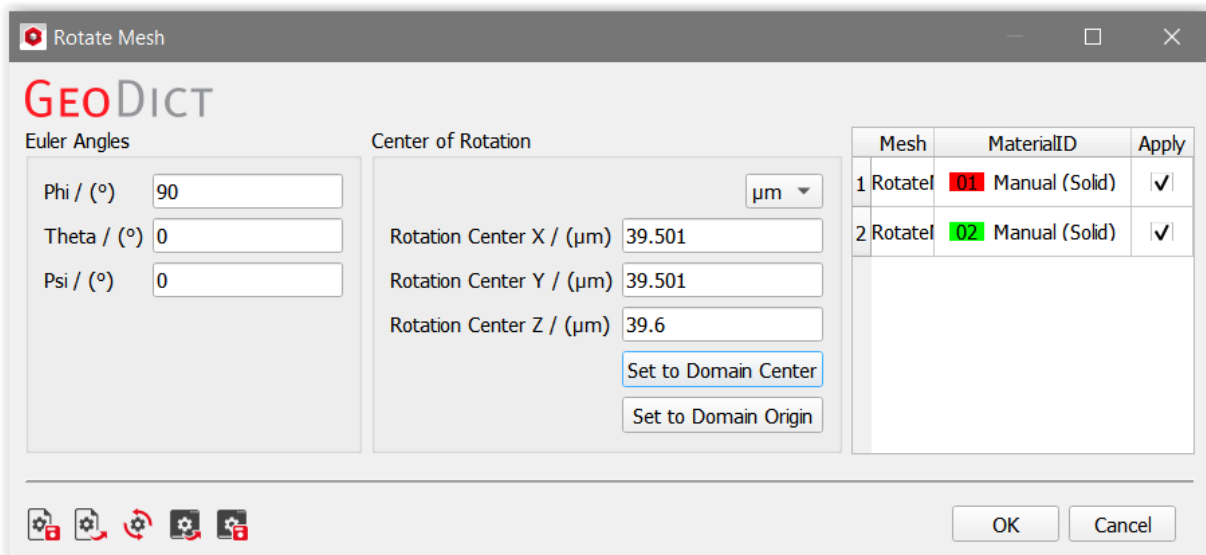
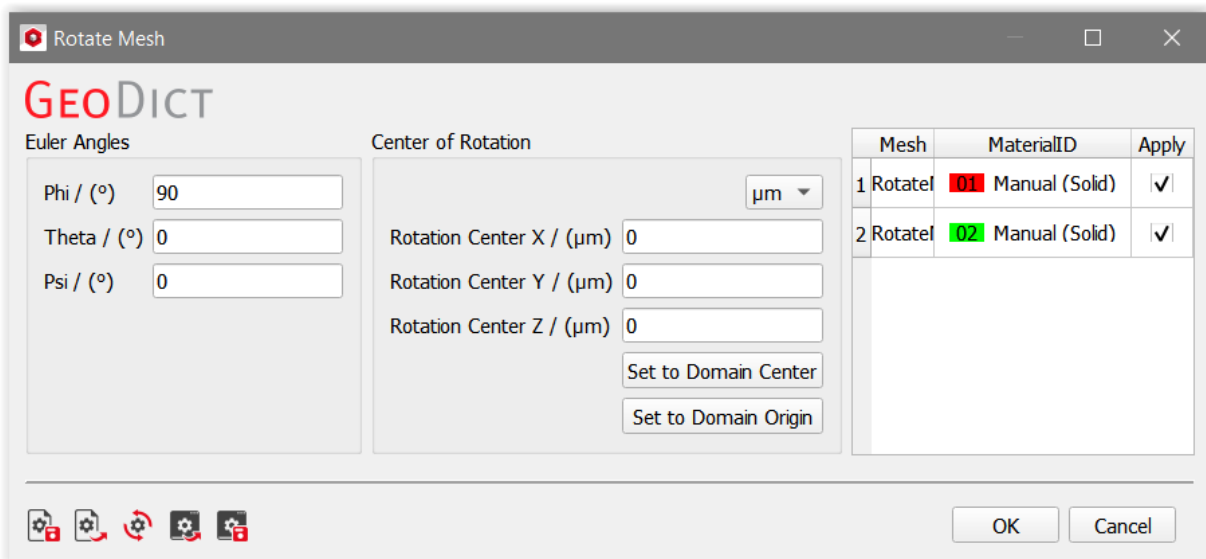
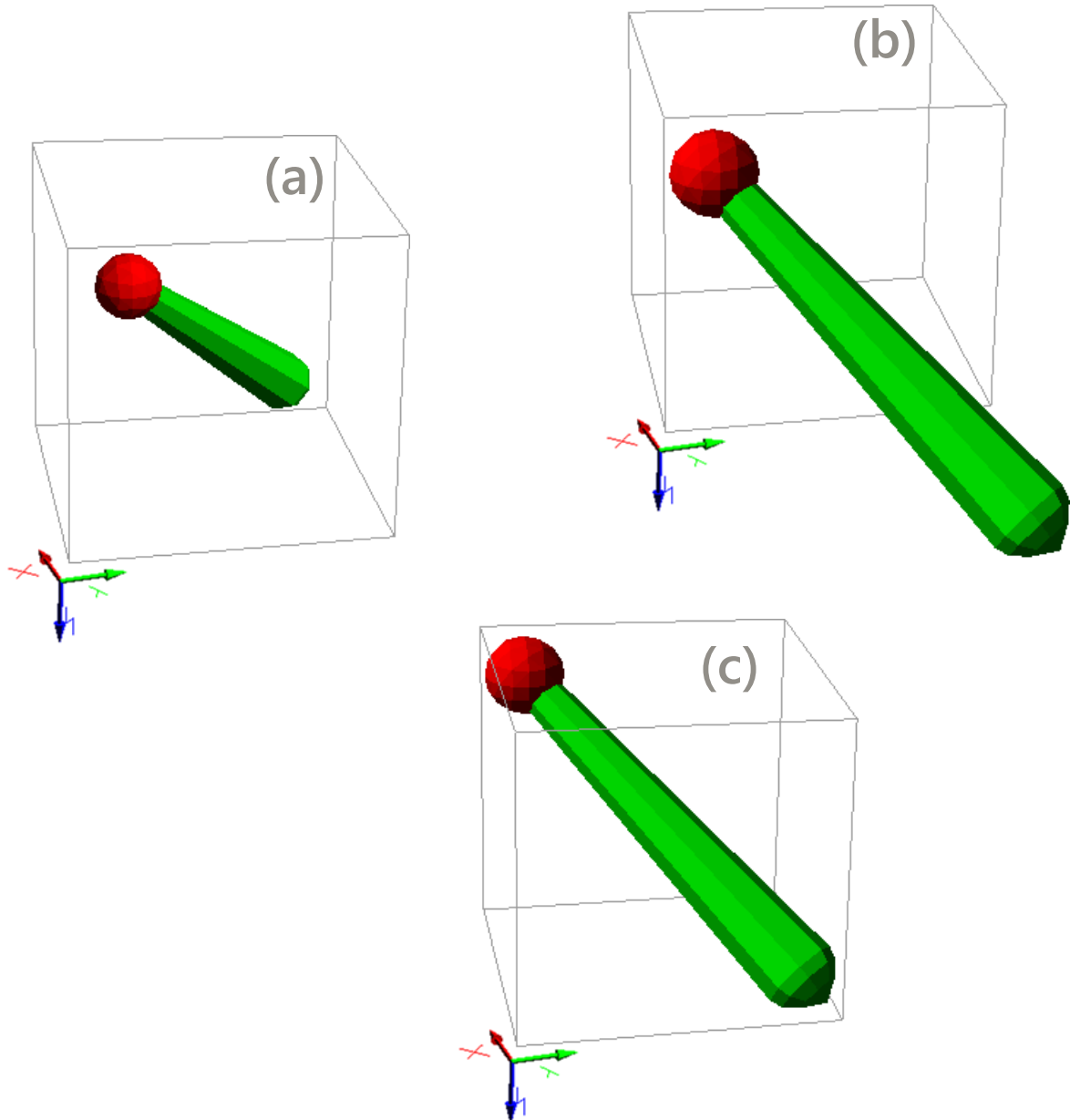


Image (a) shows the original mesh.

Image (b) shows the mesh after the rotation of Phi with  $90^\circ$  around the domain origin.

Image (c) shows the rotation of Phi with  $90^\circ$  around the domain center.



Technical  
documentation:

**Sebastian Rief**

**Liping Cheng**

**Andreas Grießer**

**Barbara Planas**

**MATH**  
**2 MARKET**

Math2Market GmbH

Richard-Wagner-Str. 1, 67655 Kaiserslautern, Germany  
[www.geodict.com](http://www.geodict.com)