# ExportGeo-CAD with MeshGeo

User Guide

GeoDict release 2024

Published: March 13, 2024

GeoDict

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

Citation:
Liping Cheng, Sebastian Rief, Barbara Planas. GeoDict 2024 User Guide. ExportGeo-CAD with MeshGeo handbook. Math2Market GmbH, Germany, doi.org/10.30423/userguide.geodict



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
Web: www.math2market.de

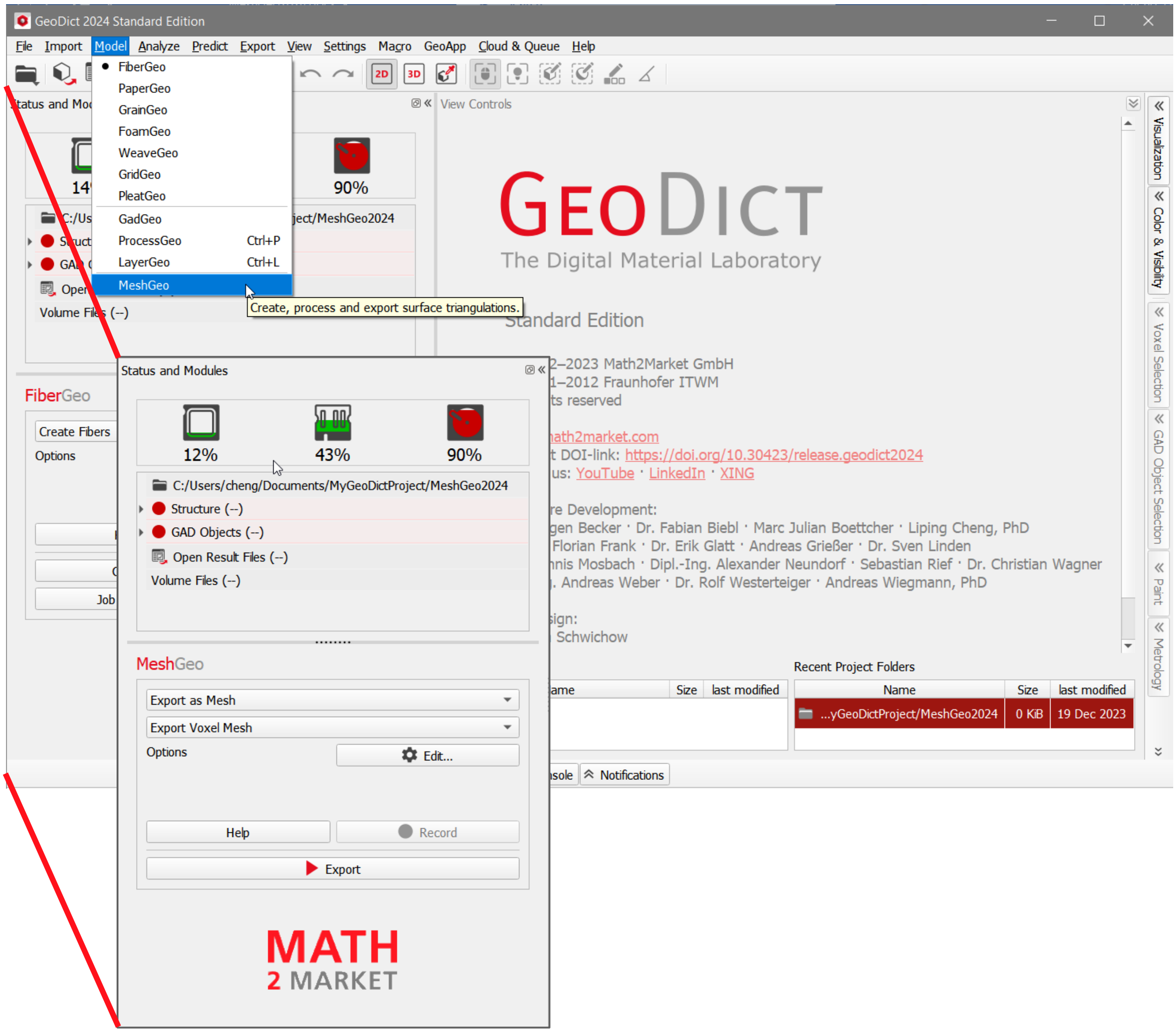
# MeshGeo / ExportGeo-CAD

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 µCT 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.

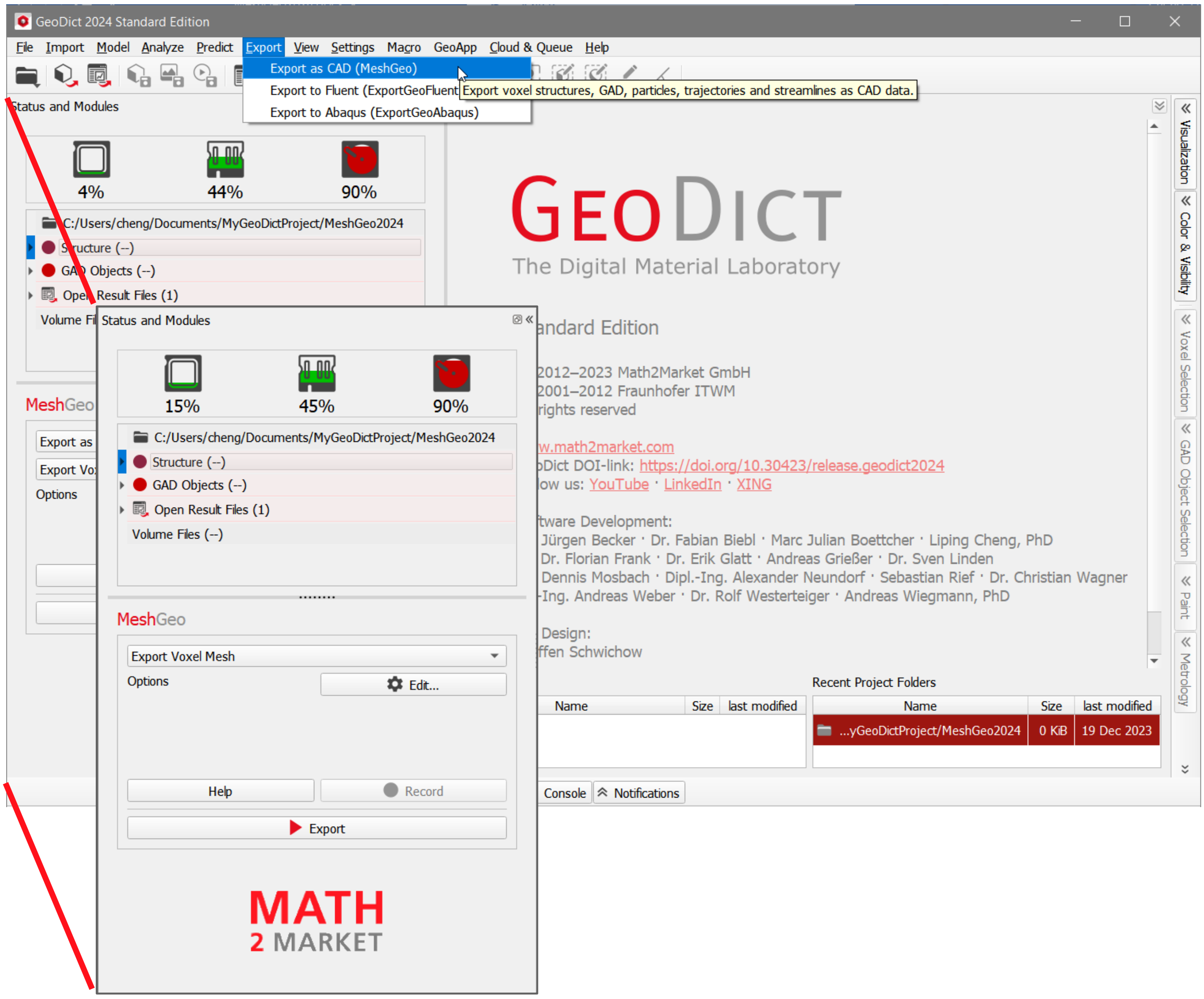
- MeshGeo can keep the mesh in memory after creation and allows subsequent smoothing and coarsening operations. For this, MeshGeo contains the additional options, **Analyze**, **Edit**, **Smooth, Coarsen & Remesh**, **Repair Mesh**, and **Voxelize Mesh**. Afterwards, the mesh can be saved in different 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:



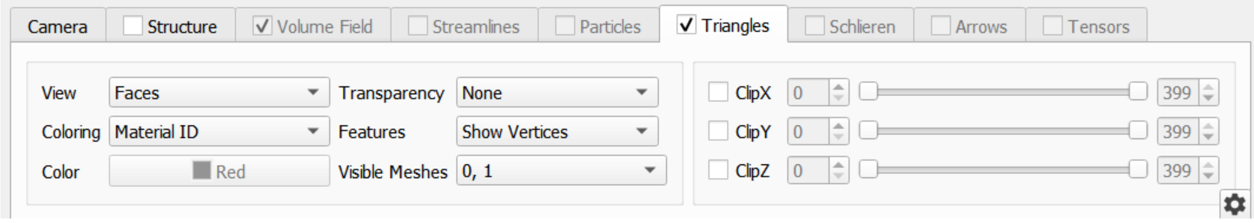
In the **MeshGeo** or **ExportGeo-CAD** section, to the left of the Visualization area, a pull-down menu displays the choices for the meshing options. The options available for **Create Mesh** are the same for both modules.

Additionally, MeshGeo contains options to further edit and save the mesh. For ExportGeo-CAD, the mesh can only be directly saved after the creation. The available options to save the mesh are the same for both modules and explained together in pages 20.

# 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.



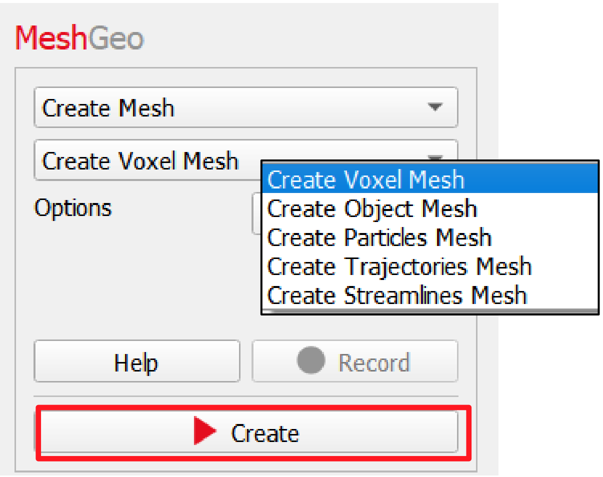
## Create Mesh

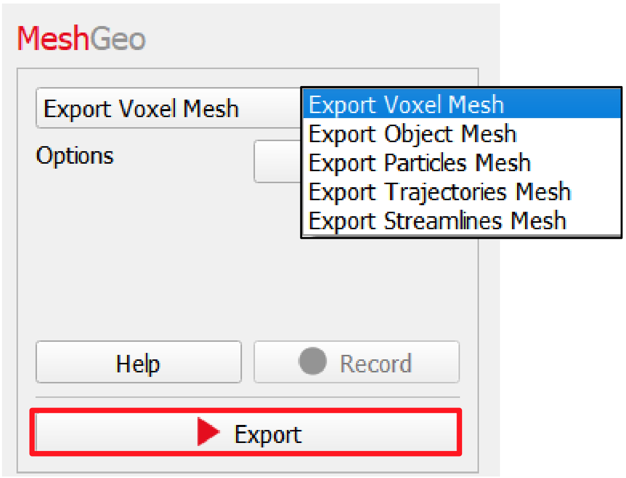
Select **Create Mesh** or **Export as Mesh** in MeshGeo, or open ExportGeo-CAD and specify the type of mesh from the pull-down menu. Meshes of a structure in GeoDict can be created based on the voxel information (**Voxel Mesh**). This voxel information can be a structure, a grey value image, an index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

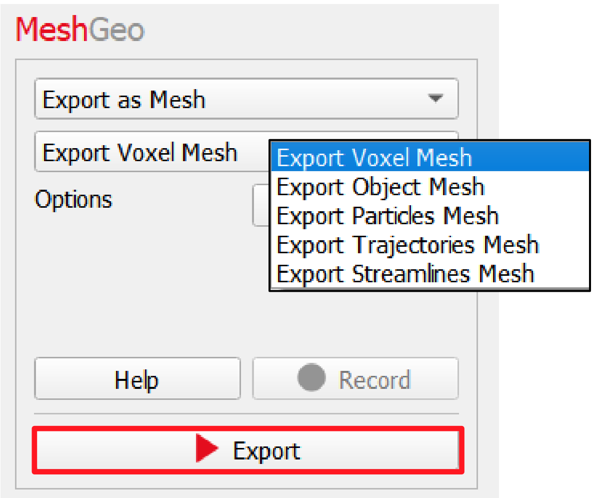
The parameters for the selected mesh type can then be defined by clicking the **Edit...** button.




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

Since GeoDict 2024, the **Export as Mesh** is added also to MeshGeo and the functionality is the same as in ExportGeo-CAD.

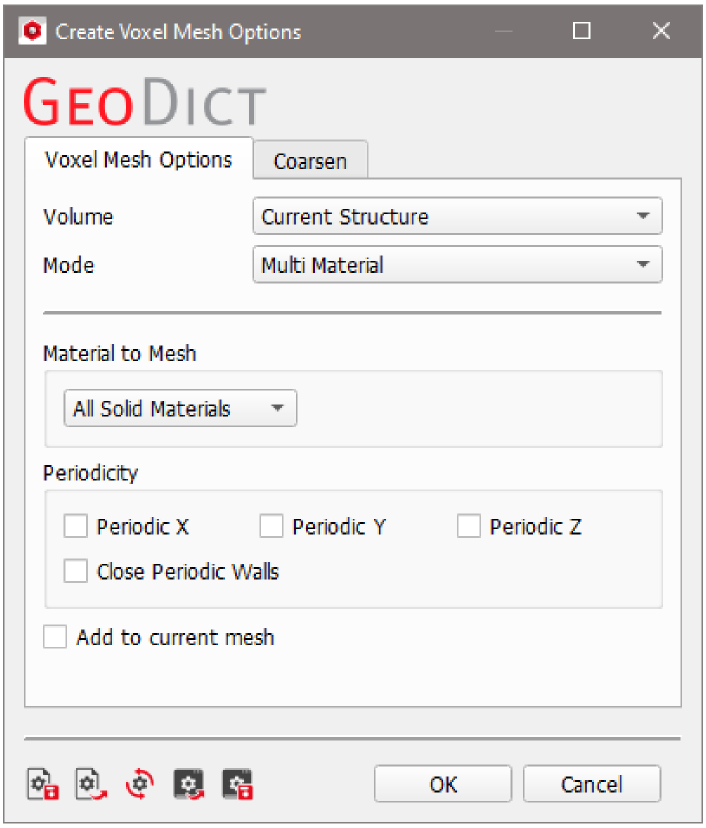


# Voxel Mesh

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 MeshGeo -> **Export Voxel Mesh** or ExportGeo-CAD since the mesh is directly saved after creation.

It contains the same options as **Save Mesh** in MeshGeo. See page 20 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 → Create Voxel Mesh** or **Export as Mesh → Export Voxel Mesh**. The available options are explained below.

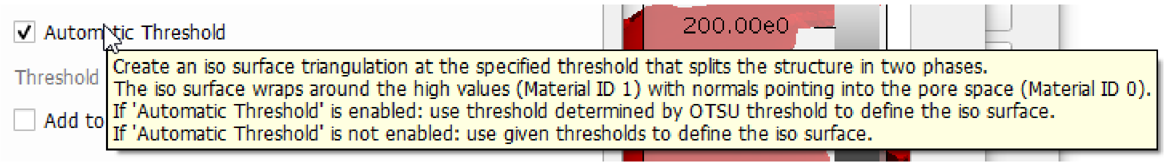
#### Current Structure

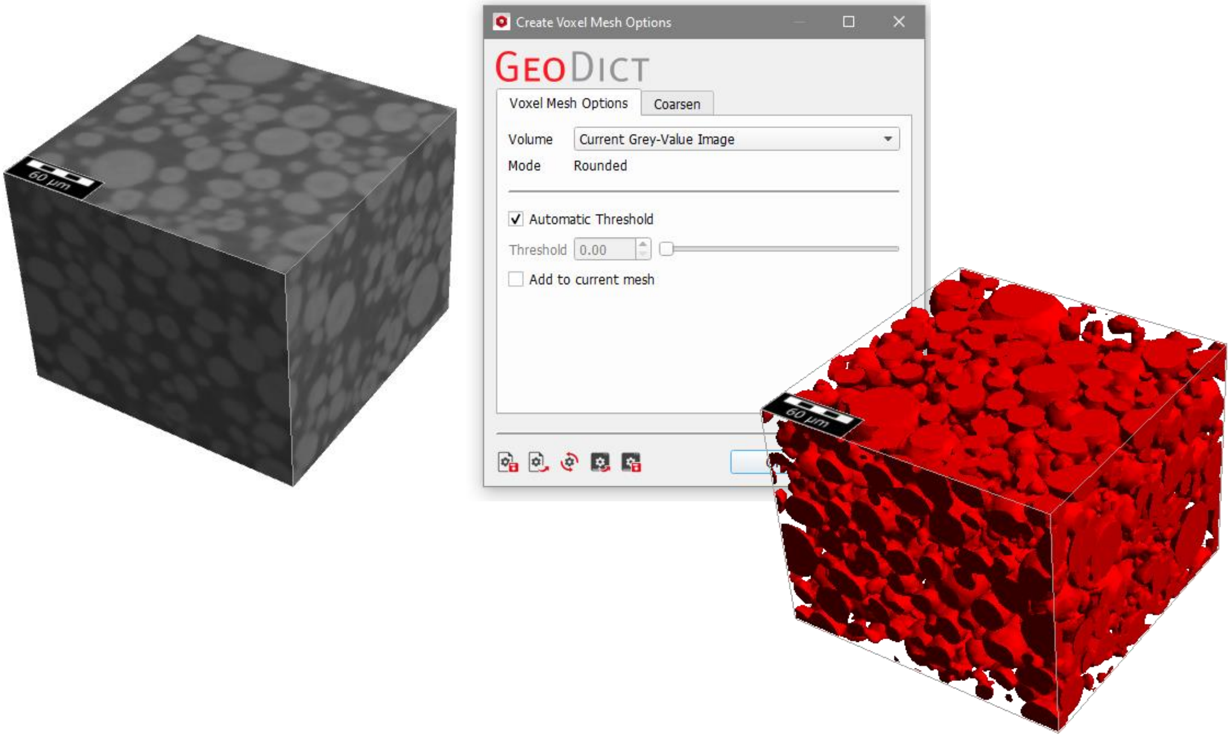
With this option, the currently loaded voxel structure is converted to a mesh. For Current Structure, all four **Mode**s are available: **Voxel Surface**, **Rounded**, **Smooth** and **Multi Material**.

#### 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 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 points into the pore space. If **Automatic Threshold** is selected, the OTSU algorithm is used to determine the threshold.

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, since the grey value information allows to obtain sub-voxel precision.

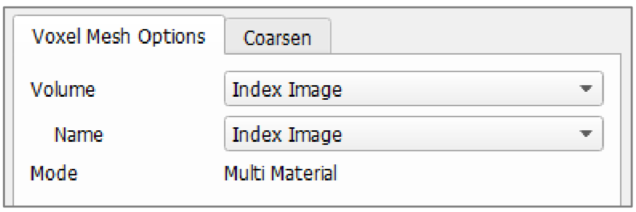


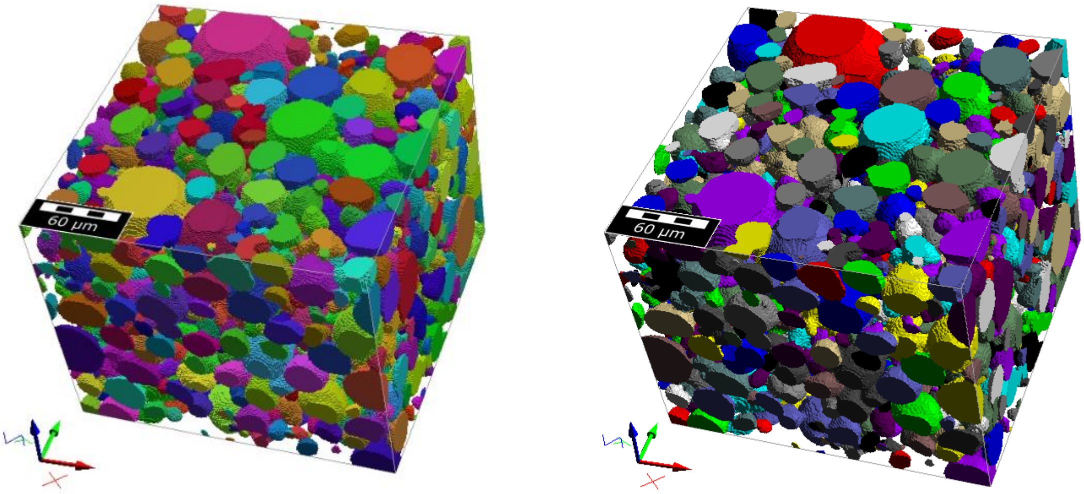

## Index Image

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.

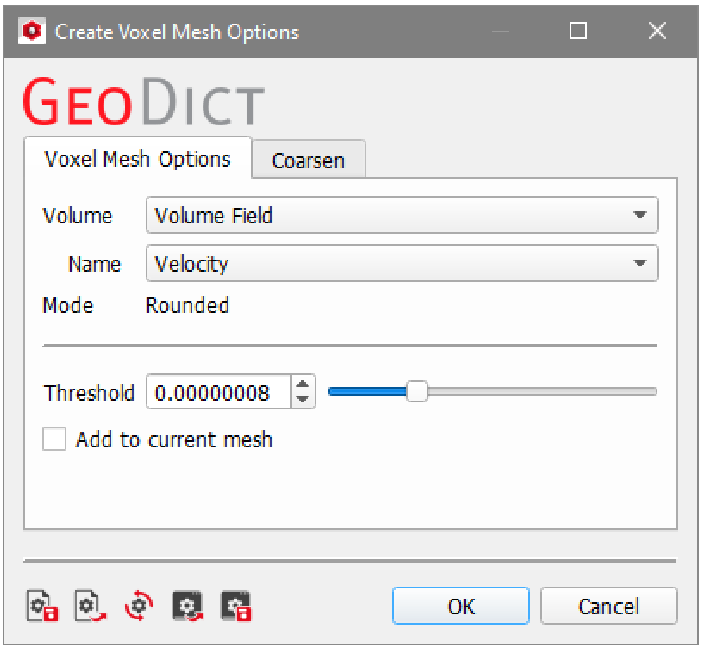


In the figure below, to the left an index image from a GrainFind result is shown. The right figure shows the mesh calculated from that index image.

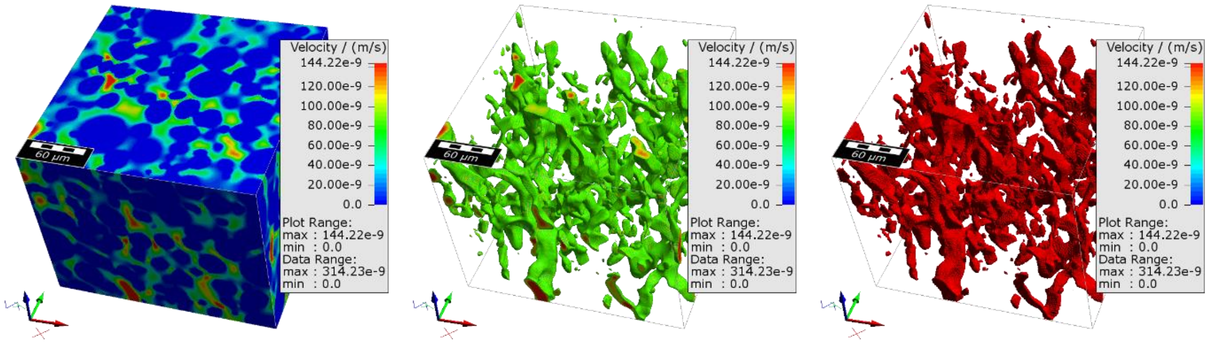



## Volume Field

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 iso-surface mesh based on the given **Threshold**.

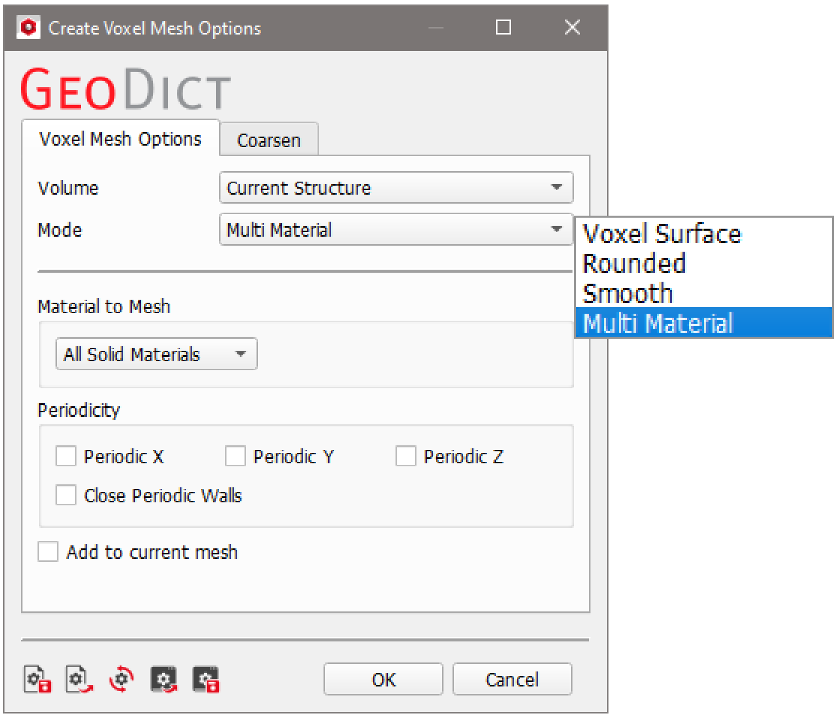



In the figure below, observe from left to right: A flow field calculated with FlowDict, the flow field with a threshold (i.e. only fast velocities are shown), and a mesh computed with the same threshold.




## MODE

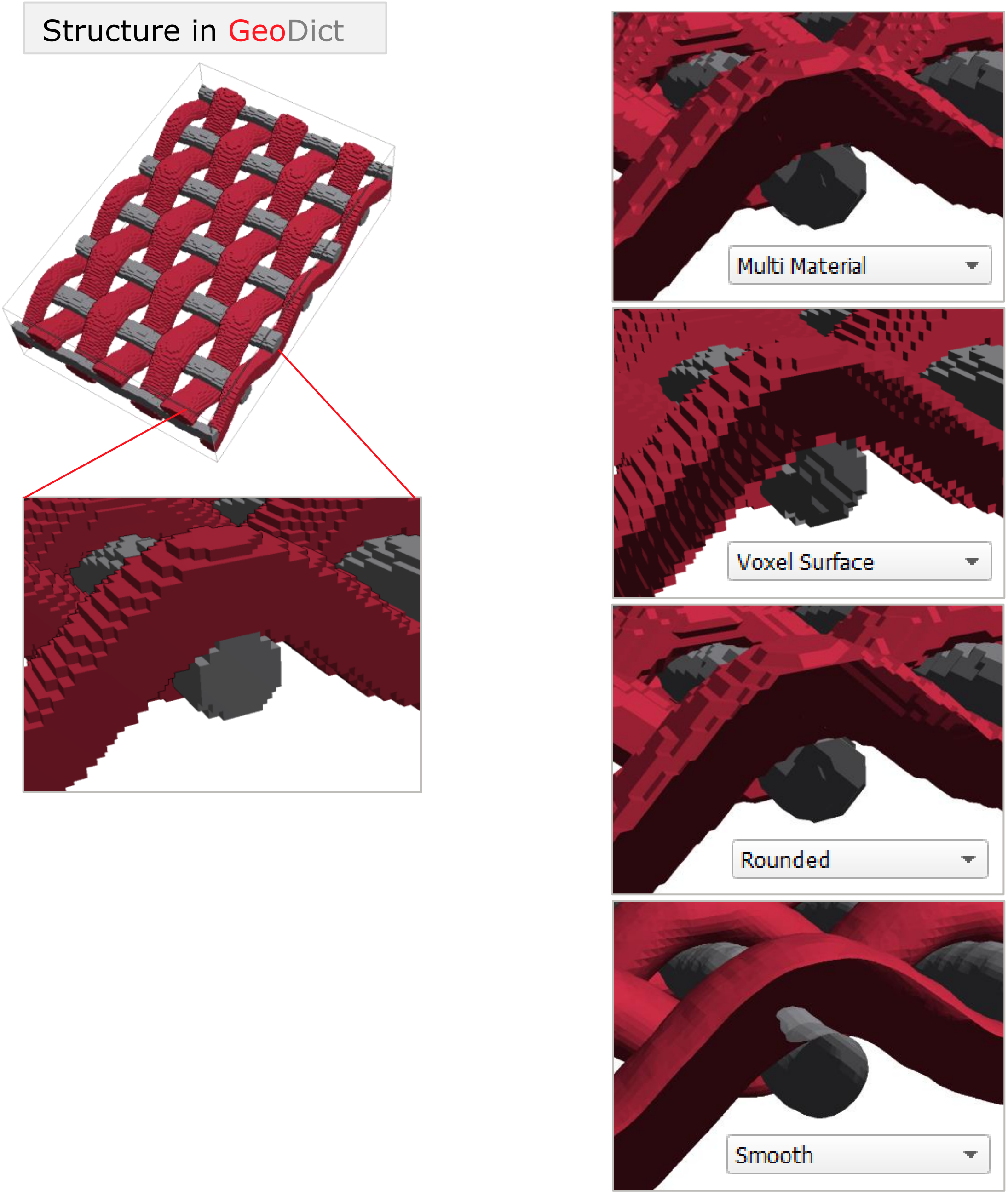
Four **Mode**s can be used to export voxel data: **Voxel Surface**, **Rounded**, **Smooth**, or **Multi Material**. **Multi Material** is the default. Which options are available depends on the selected Volume: For example, if the **Current Grey-Value Image** is selected, only **Rounded** is available.




**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. Use the **Voxel Surface** mode only if you intend to preserve the staircase-like pattern of voxel structure, otherwise, **Multi-Material** mode is recommended.

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).



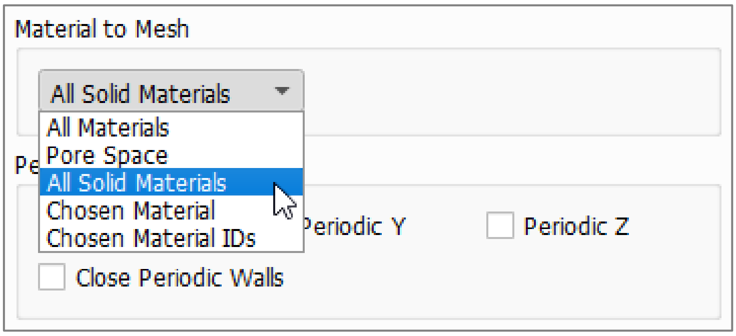

With the **Multi Material** option, the meshes for the different materials do not overlap. They have perfect contact at the points where different materials touch. When all materials are exported, a partition of unity is created. This means that each point in the domain is enclosed in exactly one surface. 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.

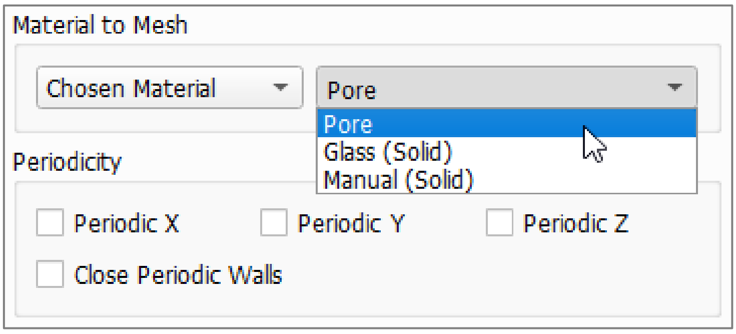
### Material to Mesh

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.

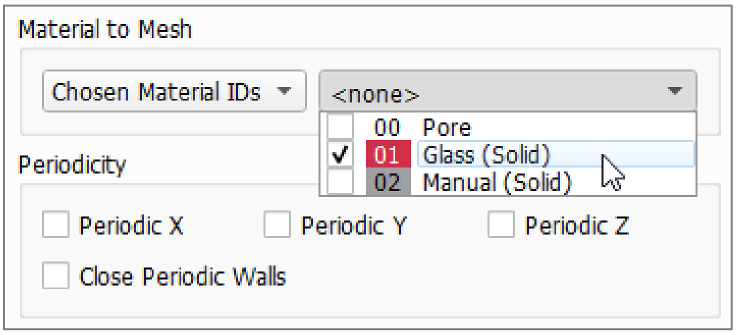
**All Solid Materials** is the default, since this is the best choice for most use cases.




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




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




### Periodicity

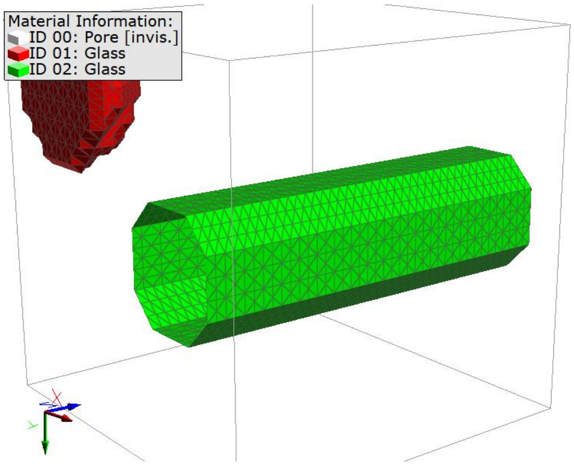
The boundary condition defines how the triangulation algorithm deals with boundary voxels.

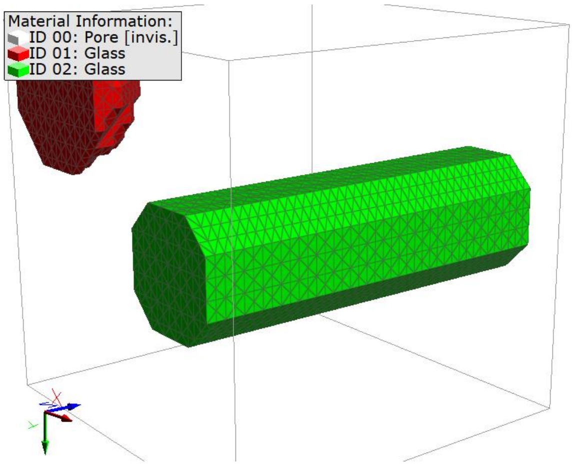
When **Periodicity** is selected, the mesh of one end is periodic to the other end of the structure in the chosen direction. This option should only be used if the original structure is periodic.

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.

The **Periodicity** 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 -> **Create Mesh**. 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.

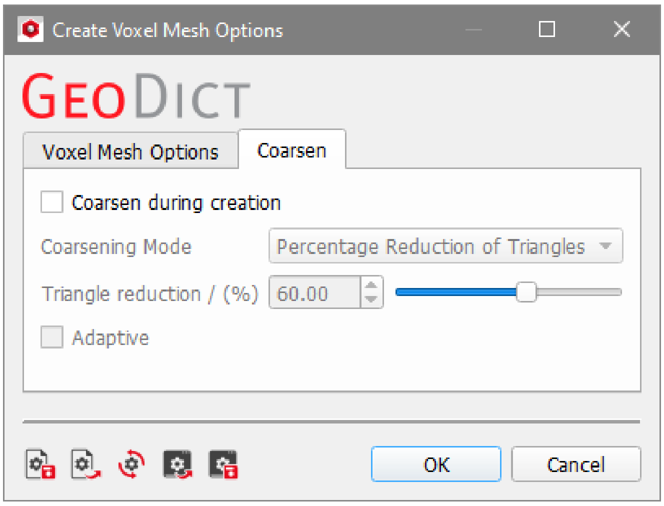
See Smooth Mesh

in page 36 and Coarsen Mesh

in page 38 .

## Coarsen

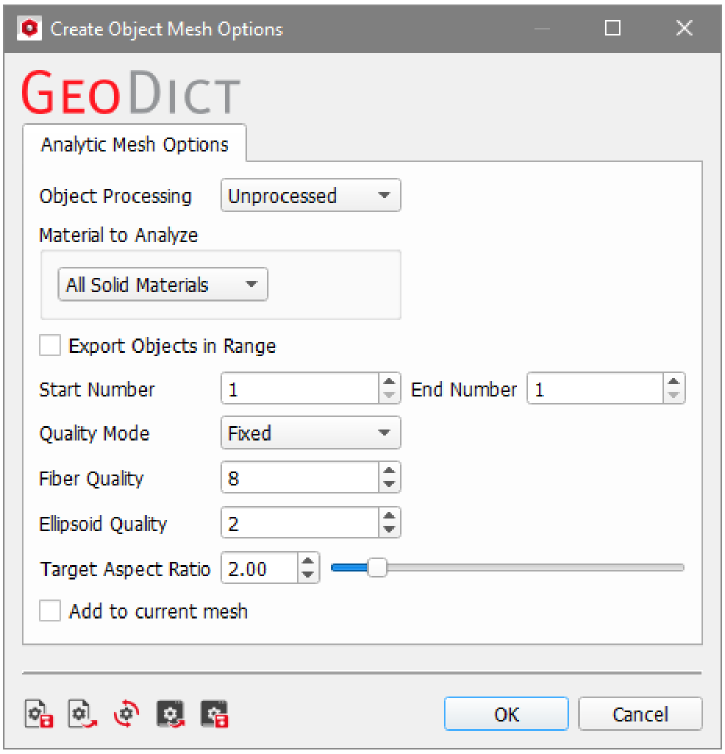
Coarsening can either be applied during mesh creation or afterwards. The settings are the same, but the results may differ – especially if the **Adaptive** mode is used. For more information, see page 38.




## Object Mesh

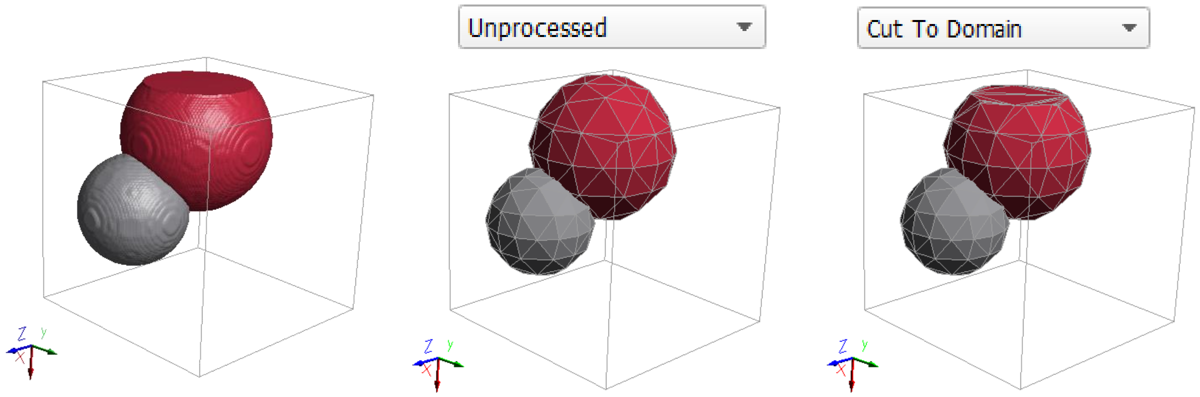
To export the objects in the structure into a mesh, select **Create Object Mesh** or **Export Object Mesh** for MeshGeo, or **Export Object Mesh** for ExportGeo-CAD from the pull-down menu. Objects in GeoDict can overlap. With **Object Mesh**, each object is meshed separately, therefore the meshes for 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.

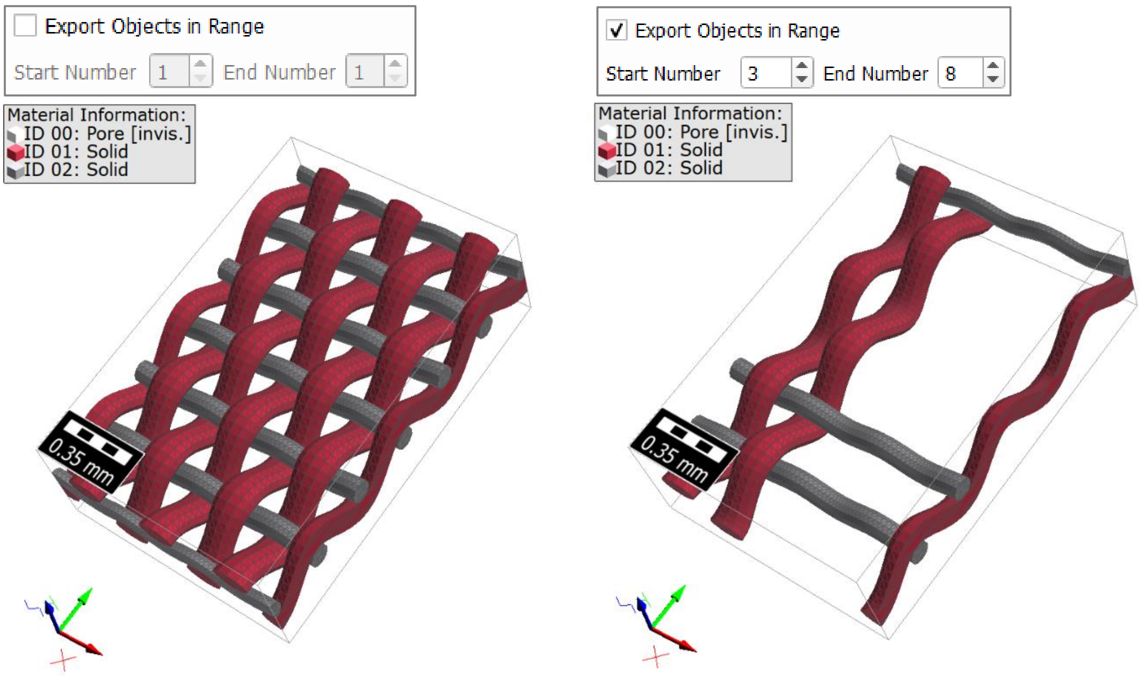



### Material to Analyze

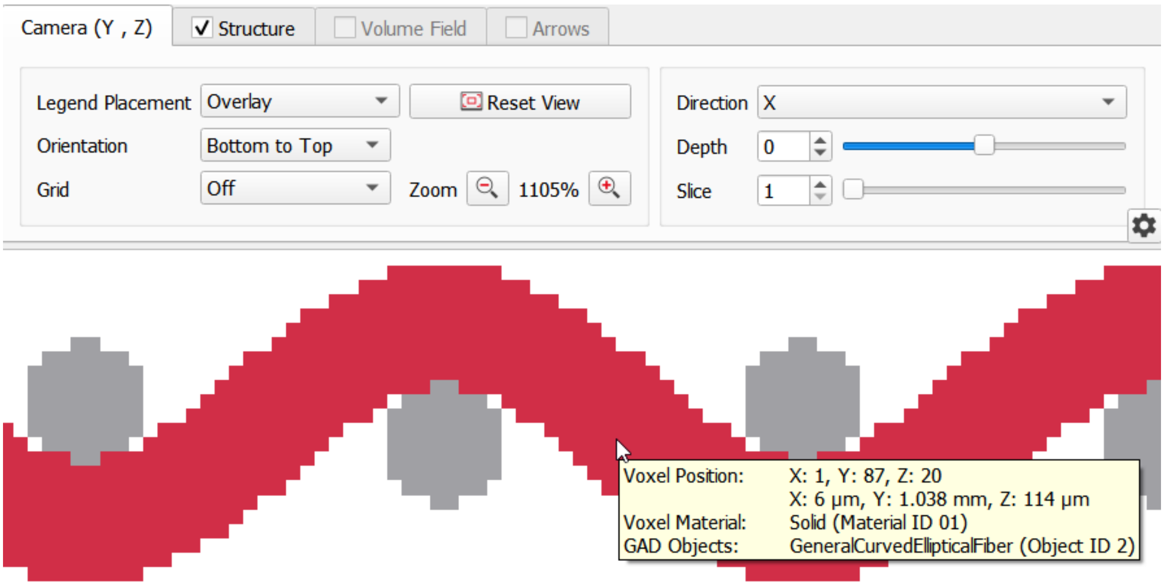
The choices for **Material to Analyze** are explained on page 9. For **Analytic Mesh**, not all material IDs in the structure can be used to create a mesh. For example, the pore space is not an object in GeoDict and therefore no analytic mesh can be created from it. Furthermore, the overlap between objects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

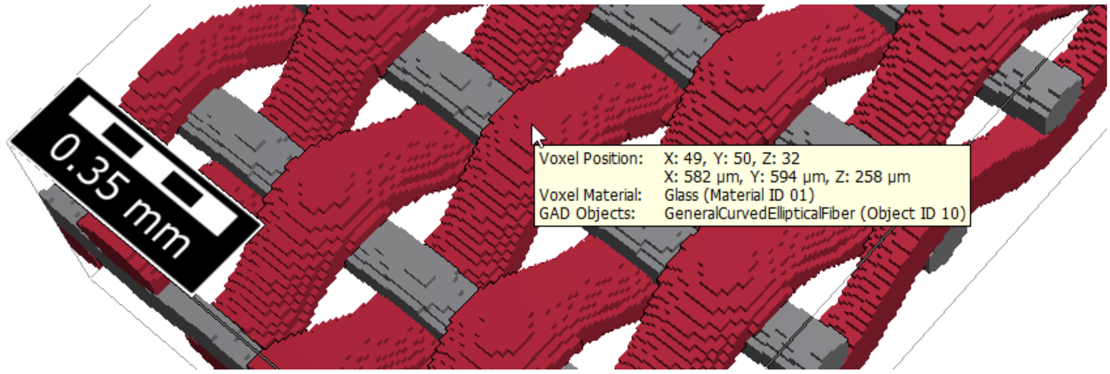
### Export Objects in Range

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.




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 (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).

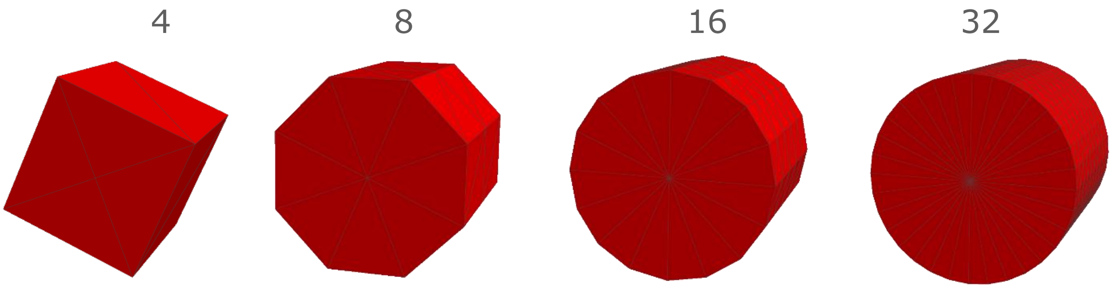
### Fiber Quality

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

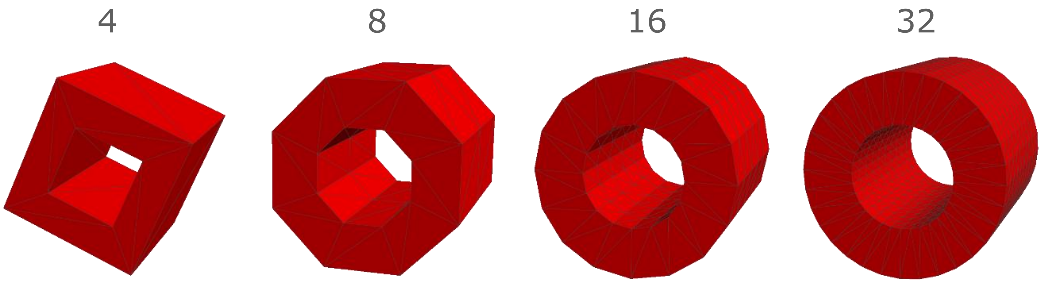
- For fibers with circular and ellipsoidal cross-section, the **Fiber Quality** corresponds to the number of nodes with which the cross-section of the fiber is discretized. This does also hold for fibers of hollow and cellulose types.

- 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.

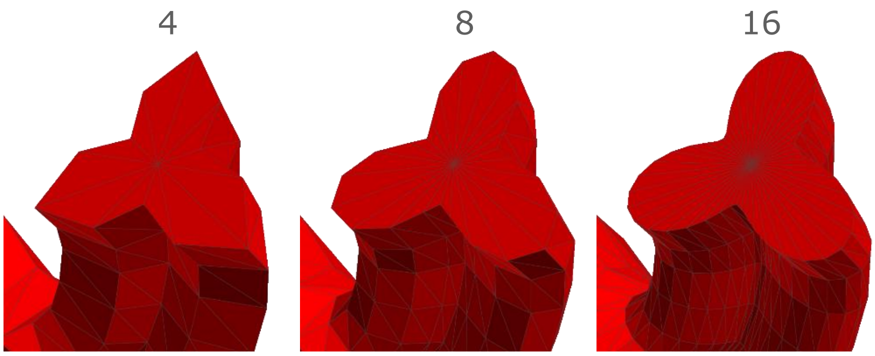
Fiber Quality for circular fibers




Fiber Quality for hollow fibers

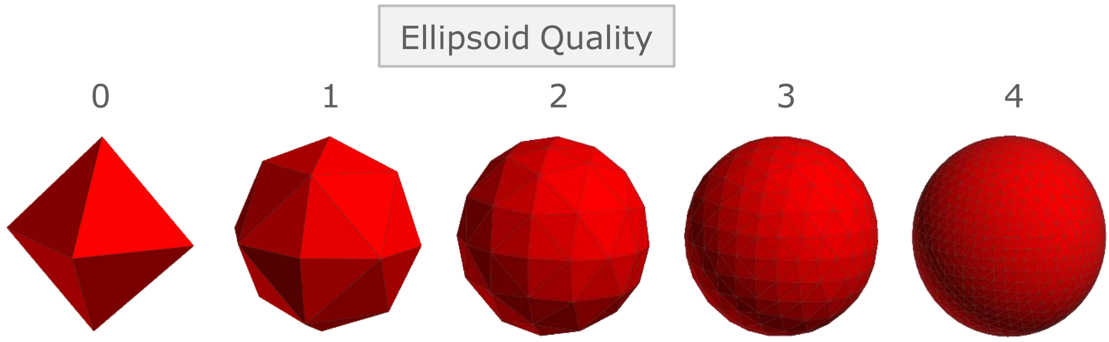



Fiber Quality for Rosetta fibers




## 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.

### Add to Current Mesh

See Add to current mesh, in page 11.

## Particles Mesh

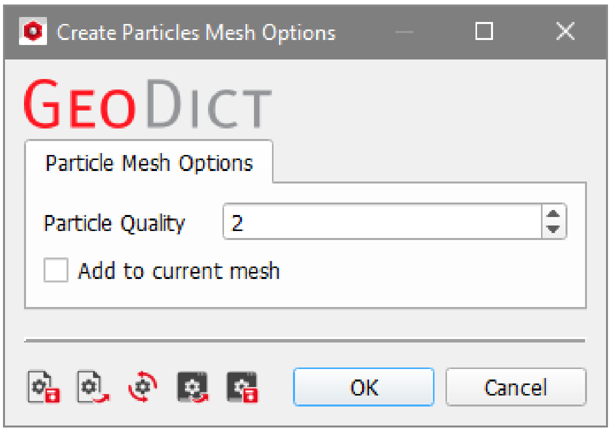
Particle positions and trajectories can be obtained from FilterDict and AddiDict simulations. MeshGeo can create a mesh from the currently loaded particles in GeoDict. The mesh is always created at the particle positions currently shown in GeoDict. Particles and trajectories can be loaded e.g. from a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

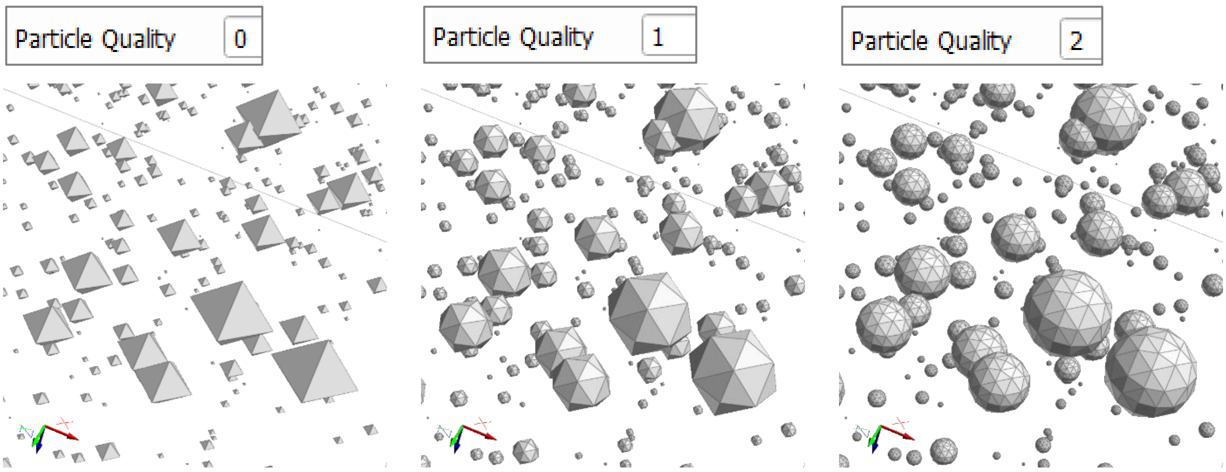
Select **Create Particles Mesh** in MeshGeo, or **Export Particles Mesh** in MeshGeo or ExportGeo-CAD and click the **Options' Edit...** button to set the parameters. When all settings are chosen, click the **Create** or **Export** button at the bottom of the MeshGeo or ExportGeo-CAD section to create 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 15).




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



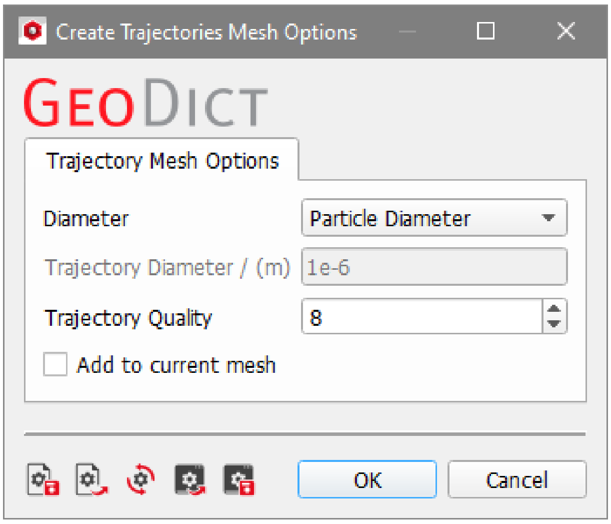
## Trajectories Mesh

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.

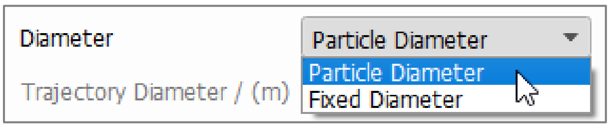
Under the **Particle Visualization** (FilterDict) or **Data Visualization** (AddiDict) tab, make sure that **Trajectories** is checked and click the **Load Particles** button.

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 **Create** 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.

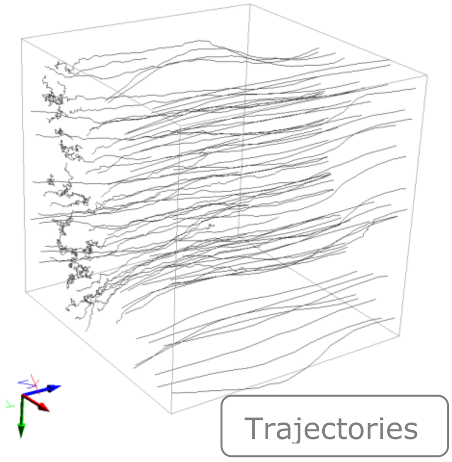


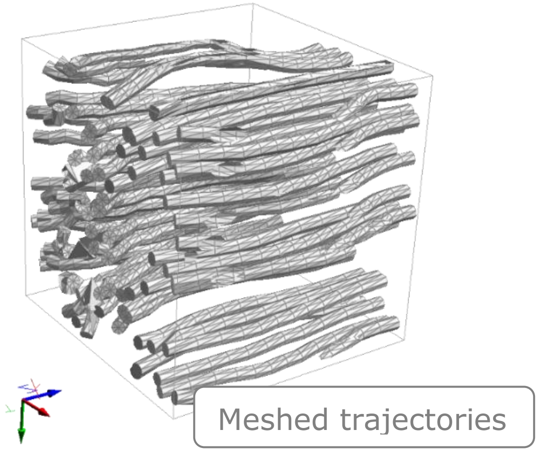

### Trajectory Quality

The **Trajectory Quality** sets the quality of the trajectory mesh. It is defined analogously to the **Fiber Quality** for analytic meshes. See page 14 for further information.

### Add to current Mesh

See Add to current mesh in page 11.

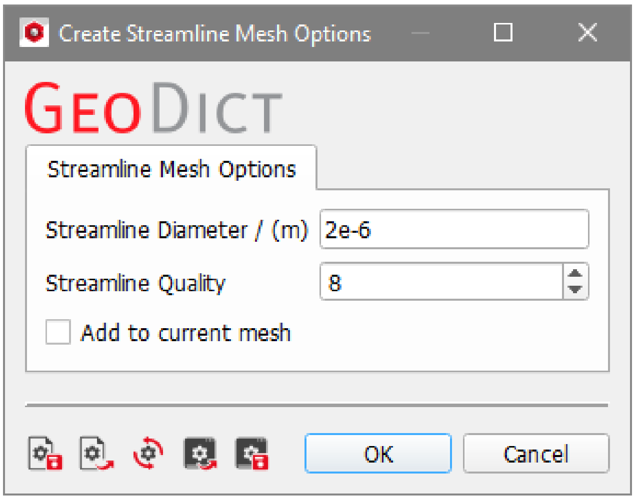

Trajectories


Meshed trajectories

## 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 **Create** button at the bottom of the MeshGeo section to create the mesh.




### Streamline Diameter

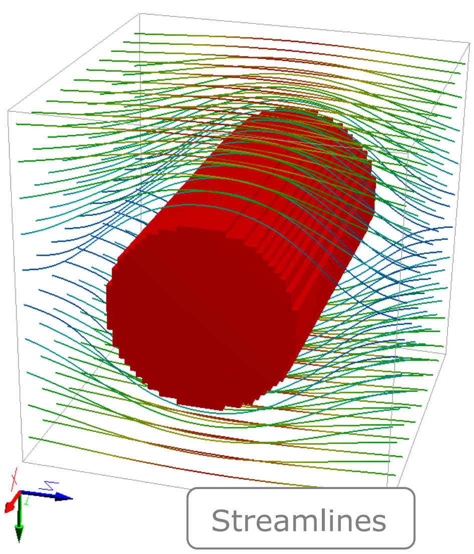
The **Streamline Diameter** option defines the diameter of the meshed streamlines.

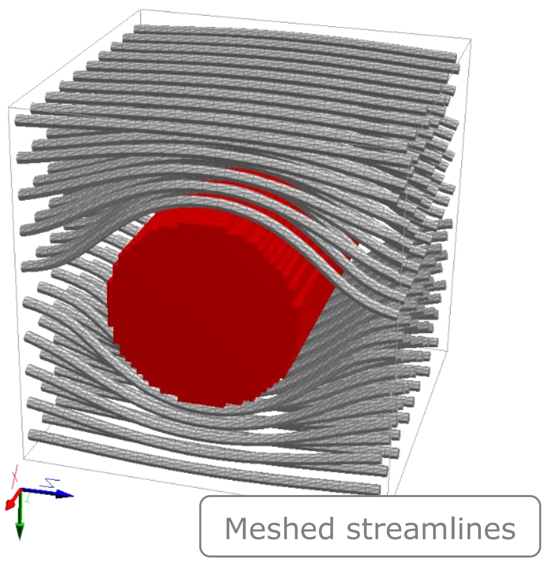
### Streamline Quality

The **Streamline Quality** sets the quality of the streamlines mesh. It is defined analogously to the Fiber Quality for round fibers (In **Create Mesh** → **Create Analytic Mesh**). See page 14 for further information.

### Add to current mesh

See Add to current mesh in page 11.



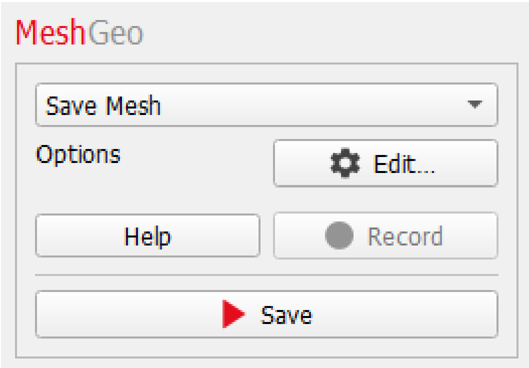



# Save Mesh

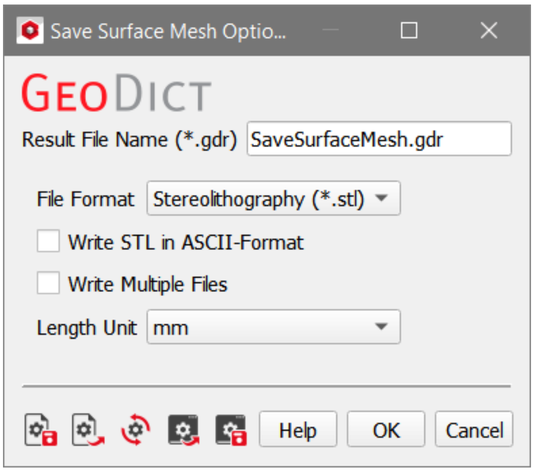
**Save Mesh** writes the currently loaded mesh to a file. When a surface triangulation is loaded, select **Save Mesh,** and then click **Save**.

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).

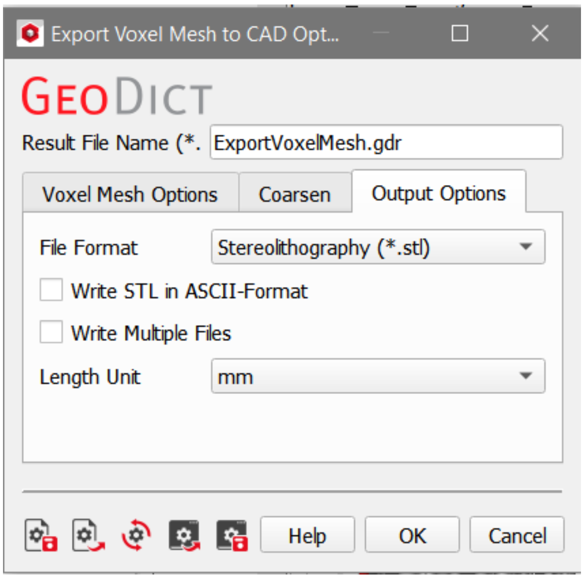
**Save Mesh** can also be reached via **File** → **Save Triangulation as…**.




**Save Mesh** options in MeshGeo




**Output Options** tab in ExportGeo-CAD or MeshGeo -> Export Mesh




## Result File Name

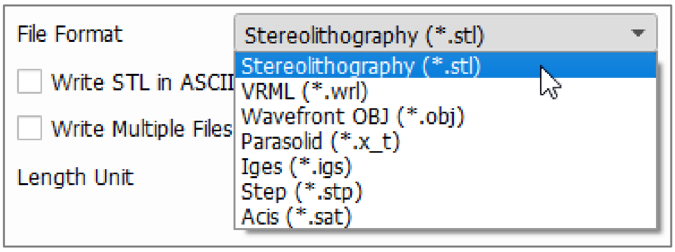
When saving a mesh, a result file is saved in the project folder and contains information about the selected parameters. Select the **Result File Name (*.gdr)** in the text box. The mesh files are also saved in the project folder.

## 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.

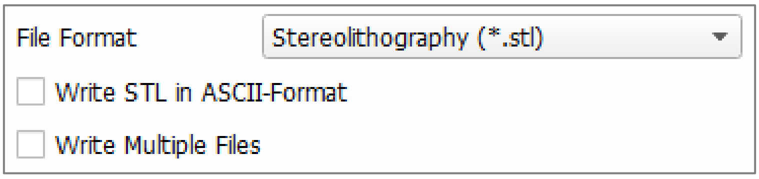



## Additional options for the *.stl format

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.

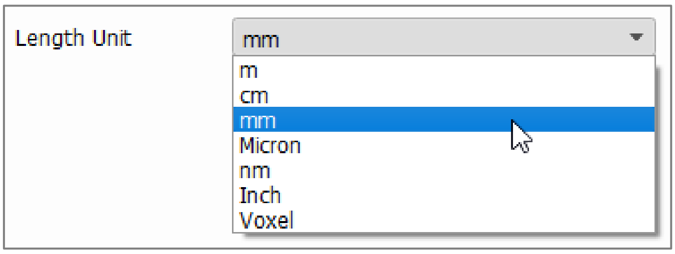



### Write Multiple Files

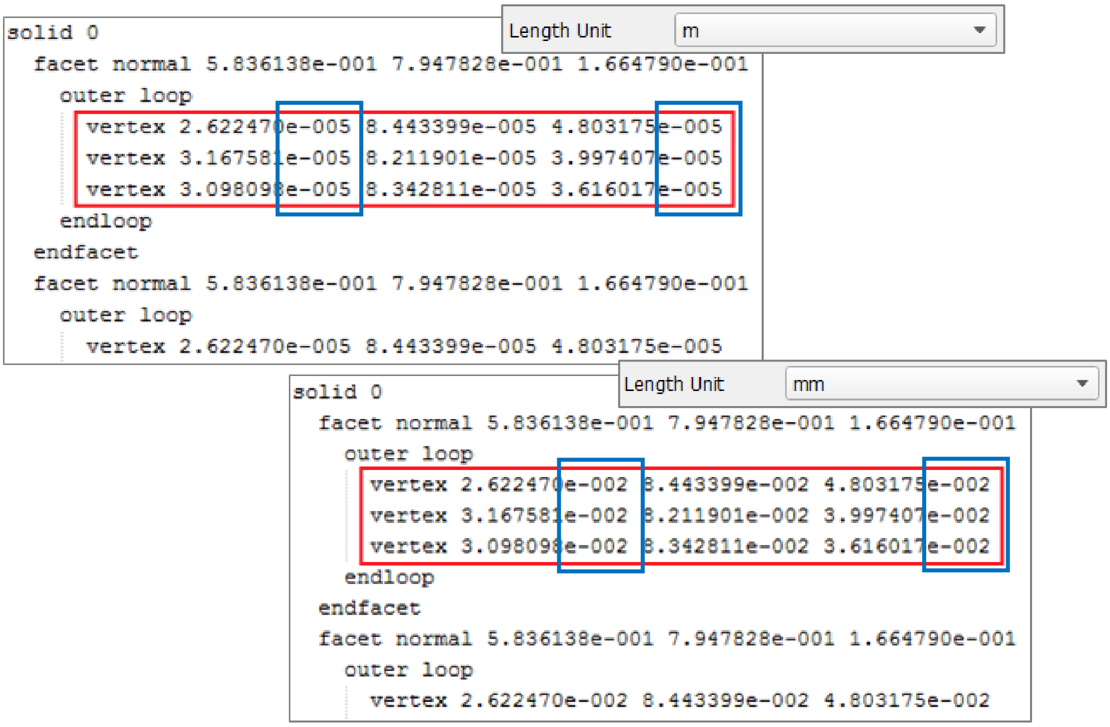
When the STL format is chosen and multiple meshes are currently loaded, these meshes can be saved to different files if necessary. This option is enabled with **Write Multiple Files**. For the other file formats, multiple meshes are always saved to one file.

## 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.

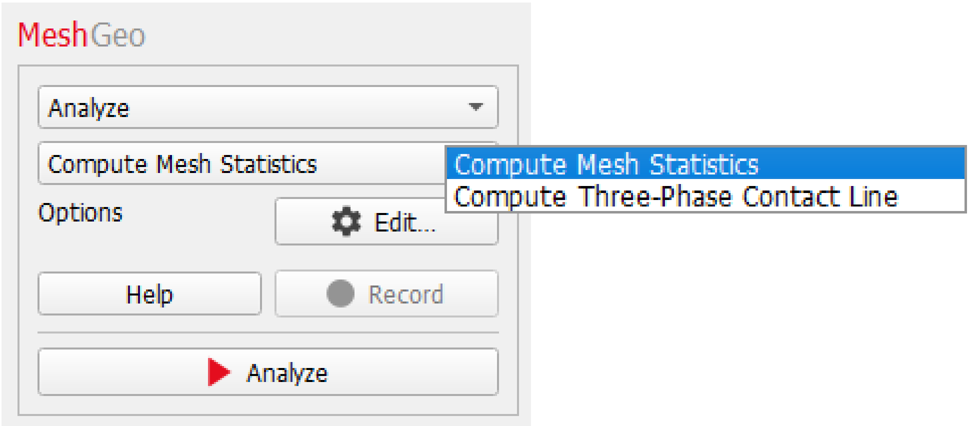



In the STL format, the information about the used unit is not contained in the file. The files for different units differ only in the order of magnitude of the data. Therefore, the unit should be chosen carefully based on your needs.


Length Unit
m
solid 0
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
Length Unit
mm
solid 0
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

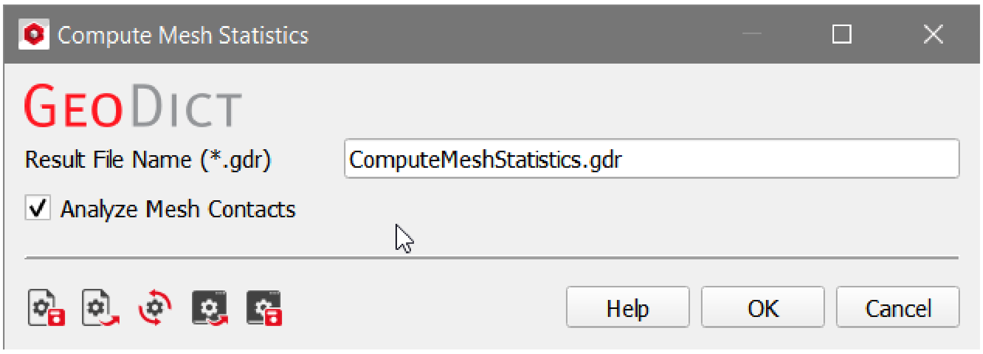
## Analyze

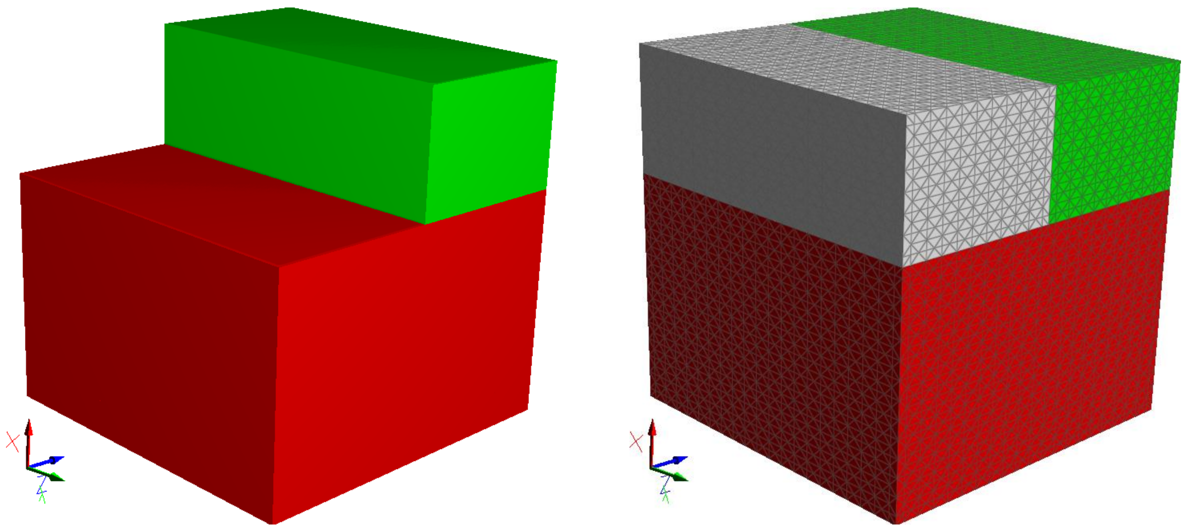
By visualizing the **Triangles**, the overall quality of the surface mesh can be determined. However, visual inspection does not allow to check for small details, and it makes it hard to check each single triangle. Since GeoDict 2024, the the mesh quality can be assessed by using the **Compute Mesh Statistics** and **Compute Three-phase Contact Line** in **Analyze**.



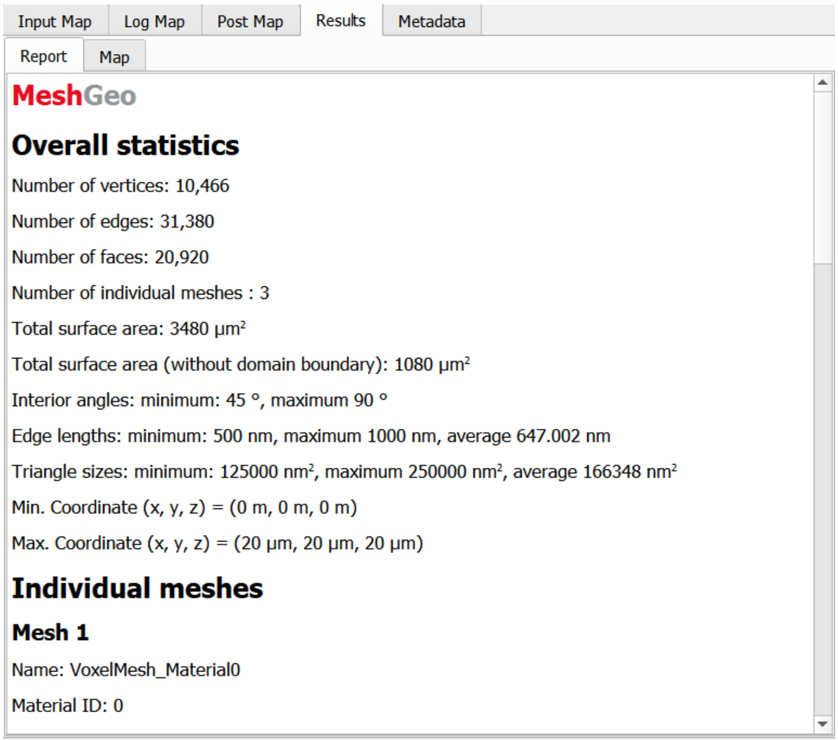

## Compute Mesh Statistics

Click the **Edit...** button to set the parameters. The option **Analyze Mesh Contacts** determines if the shared interface between individual meshes should be analyzed or not.

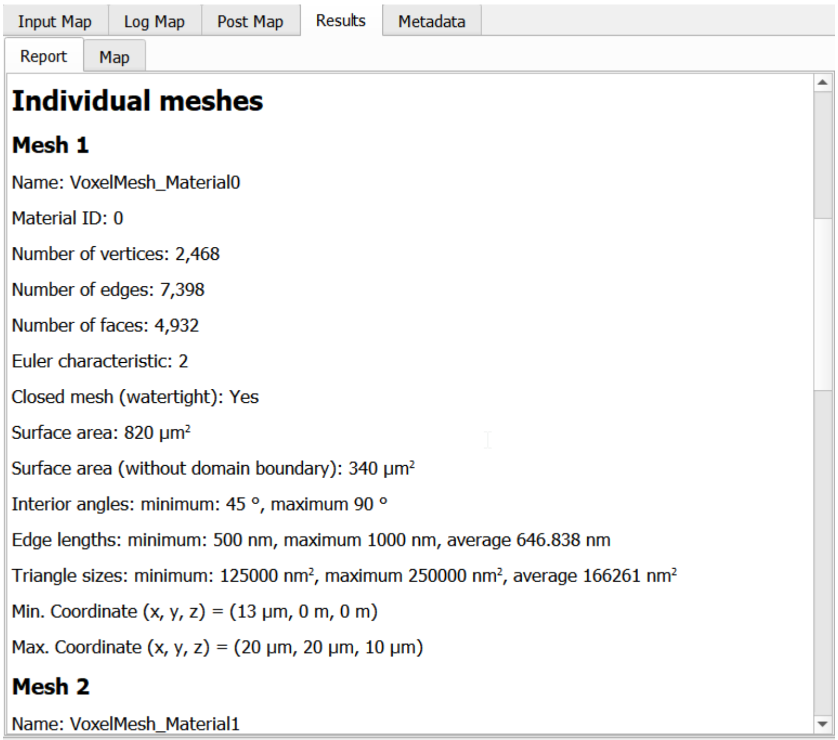


For a simple brick structure, the voxel mesh is generated for all the materials. When **Analyze** is done, the information of **Overall statistics** as well as the **Individual meshes** is given. The overall statistics include the number of vertices, edges, faces, meshes, surface area, angle, and coordinates.

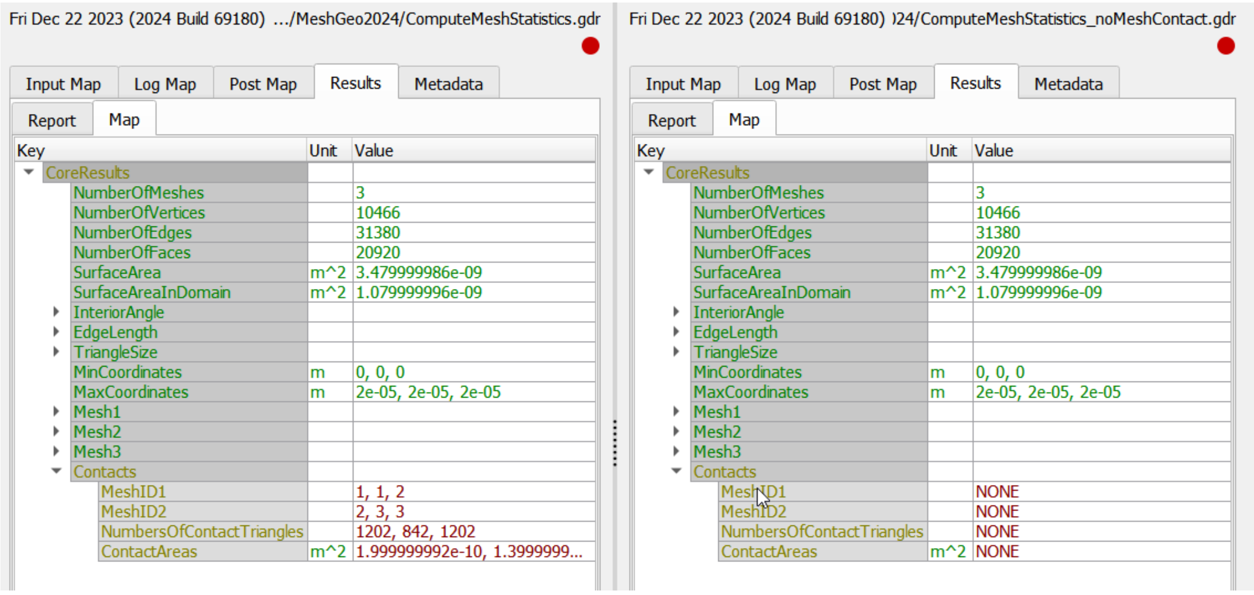



Detailed information is also given for each individual mesh.



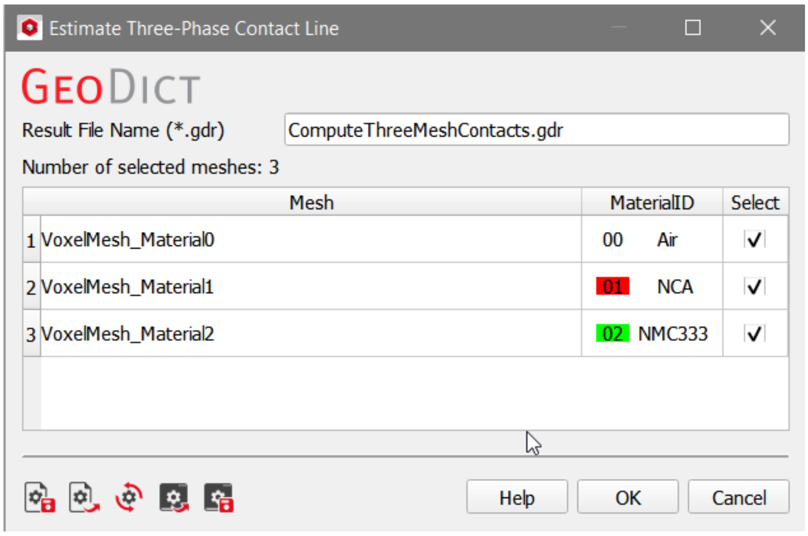

All the information can be found additionally in the result **Map** tab that can be accessed also with GeoPy scripts.

See the Result Viewer handbook of the User Guide for more information. When **Analyze Mesh Contacts** is not chosen, the information of **Contacts** is not provided.


Fri Dec 22 2023 (2024 Build 69180) .../MeshGeo2024/ComputeMeshStatistics.gdr
Input Map
Log Map
Post Map
Results
Metadata
Report
Map
Key
Unit
Value
CoreResults
NumberOfMeshes
3
NumberOfVertices
10466
NumberOfEdges
31380
NumberOfFaces
20920
SurfaceArea
m^2
3.479999986e-09
SurfaceAreaInDomain
m^2
1.079999996e-09
InteriorAngle
EdgeLength
TriangleSize
MinCoordinates
m
0, 0, 0
MaxCoordinates
m
2e-05, 2e-05, 2e-05
Mesh1
Mesh2
Mesh3
Contacts
MeshID1
1, 1, 2
MeshID2
2, 3, 3
NumbersOfContactTriangles
1202, 842, 1202
ContactAreas
m^2
1.999999992e-10, 1.3999999...
Fri Dec 22 2023 (2024 Build 69180) )24/ComputeMeshStatistics_noMeshContact.gdr
Input Map
Log Map
Post Map
Results
Metadata
Report
Map
Key
Unit
Value
CoreResults
NumberOfMeshes
3
NumberOfVertices
10466
NumberOfEdges
31380
NumberOfFaces
20920
SurfaceArea
m^2
3.479999986e-09
SurfaceAreaInDomain
m^2
1.079999996e-09
InteriorAngle
EdgeLength
TriangleSize
MinCoordinates
m
0, 0, 0
MaxCoordinates
m
2e-05, 2e-05, 2e-05
Mesh1
Mesh2
Mesh3
Contacts
MeshID1
NONE
MeshID2
NONE
NumbersOfContactTriangles
NONE
ContactAreas
m^2
NONE

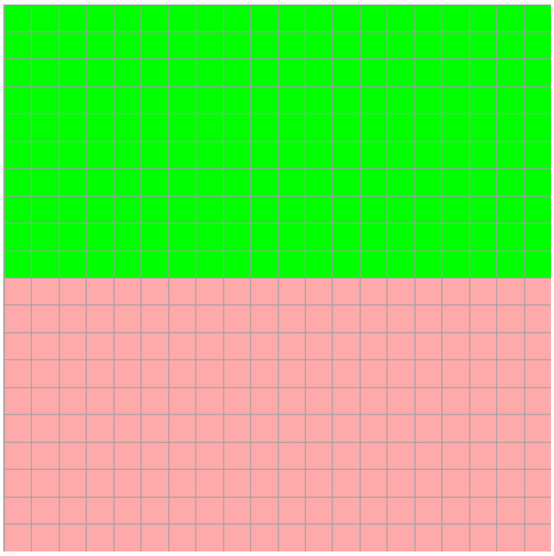
## Compute three-Phase Contact Line

To describe the interface between different materials, the length the of three-phase contact line at which three different materials meet can provide helpful information. To compute a three-phase contact line, at least 3 meshes need to be loaded. For the above brick example, we can choose the three meshes and start the computation.

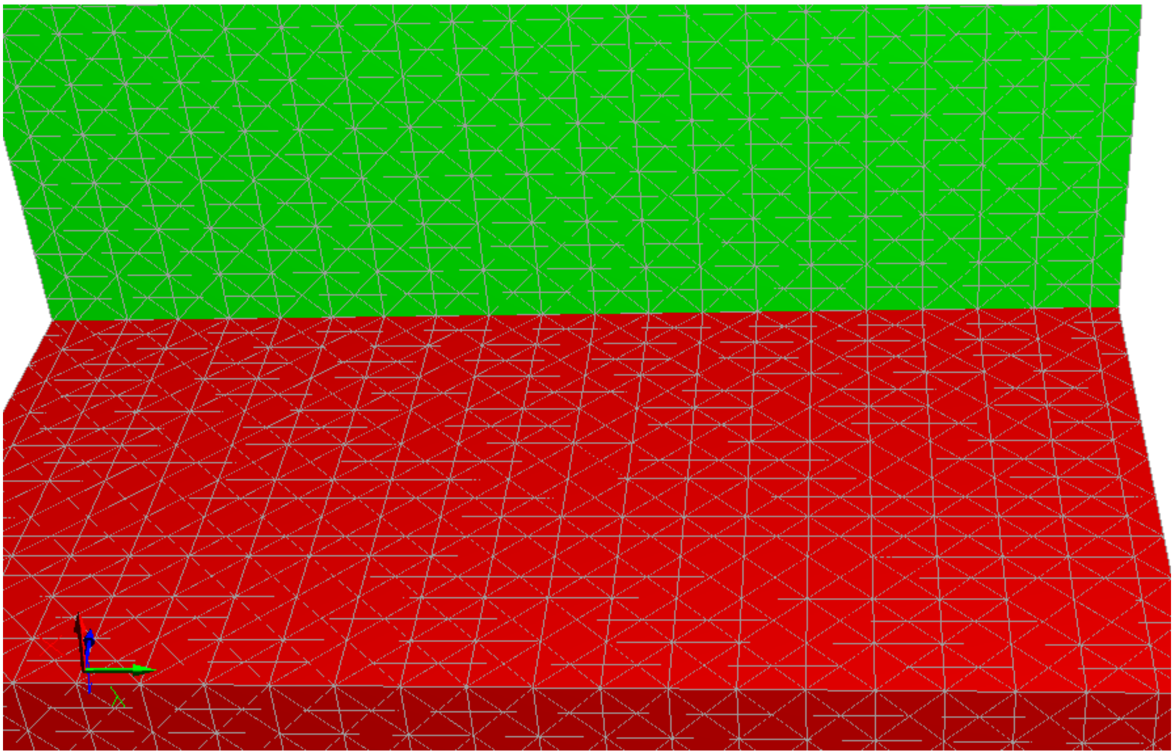



The length of the contact line and the number of edges is given.

The data can be validated easily for this small structure, which has a width of 20 voxels 20 voxels and a voxel length of 1µm.

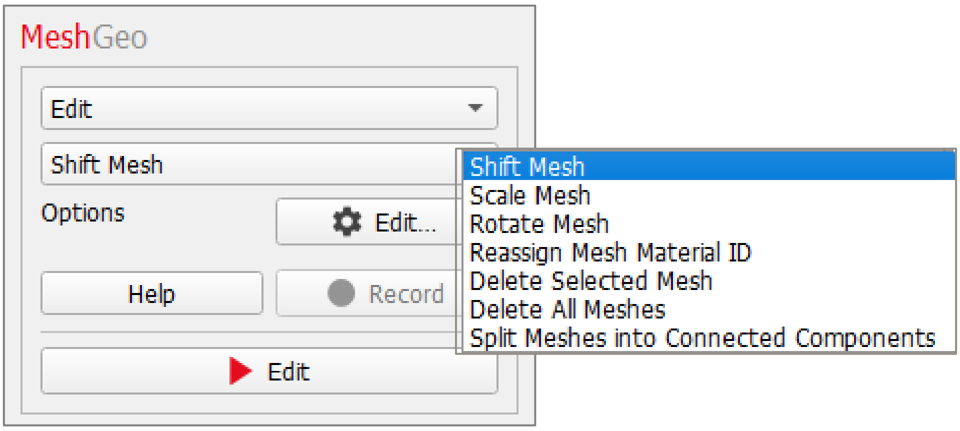


The surface mesh is shown below. As it is shown in the result viewer, we can count 21 edges, where the first and last edges of the observed contact line are half as long as the other 19 edges.



## EDIT

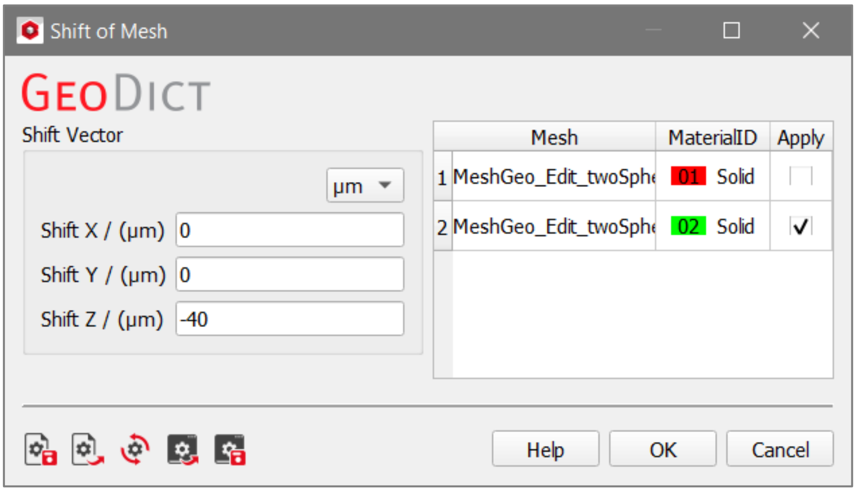
**Edit** allows modifying the loaded meshes, including **Shift Mesh**, **Scale Mesh**, **Rotate Mesh**, **Reassign Mesh Material ID**, **Delete Selected Mesh**, **Delete All Meshes**, and **Split Meshes into Connected Components**.



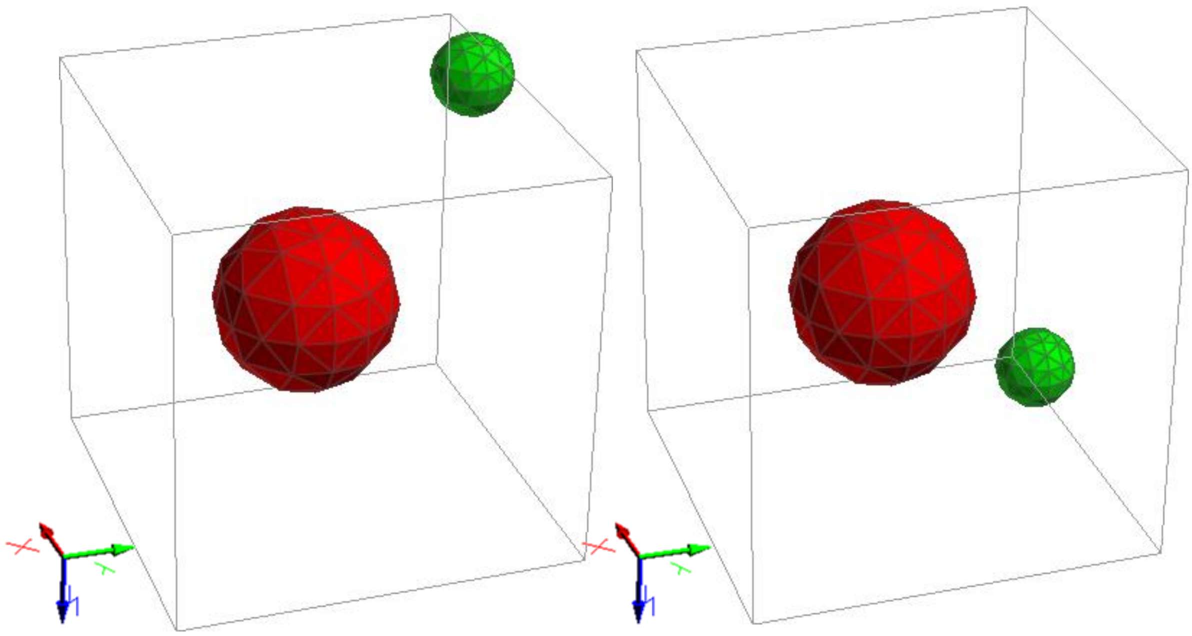
## Shift Mesh

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

Select **Shift Mesh** and click the upper **Edit...** button (to the right of **Options**) to open the **Shift Mesh** dialog. Choose the mesh to be shifted on the right box and define 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 the **Edit** button at the bottom (the one with the red arrow) to run the command.



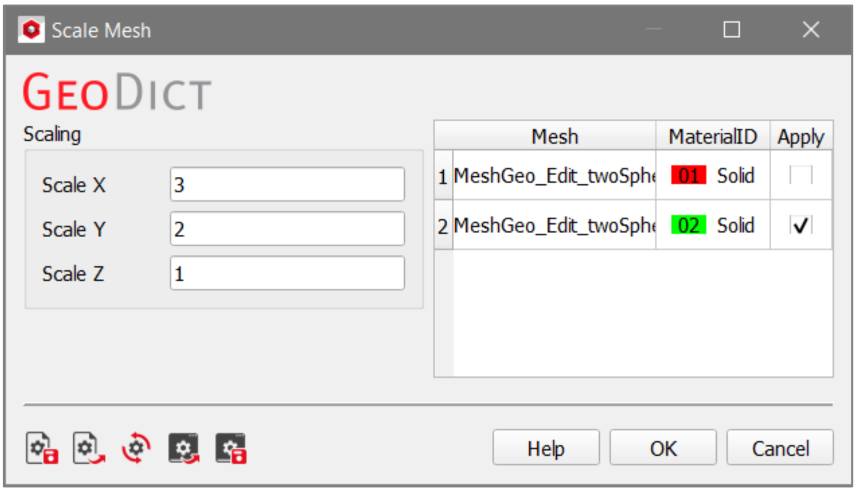

In the example below, there are two meshes, one for the big red particle and the other one for the small green particle. After shifting, the small green particle is moved in the negative Z-direction by 40µm.



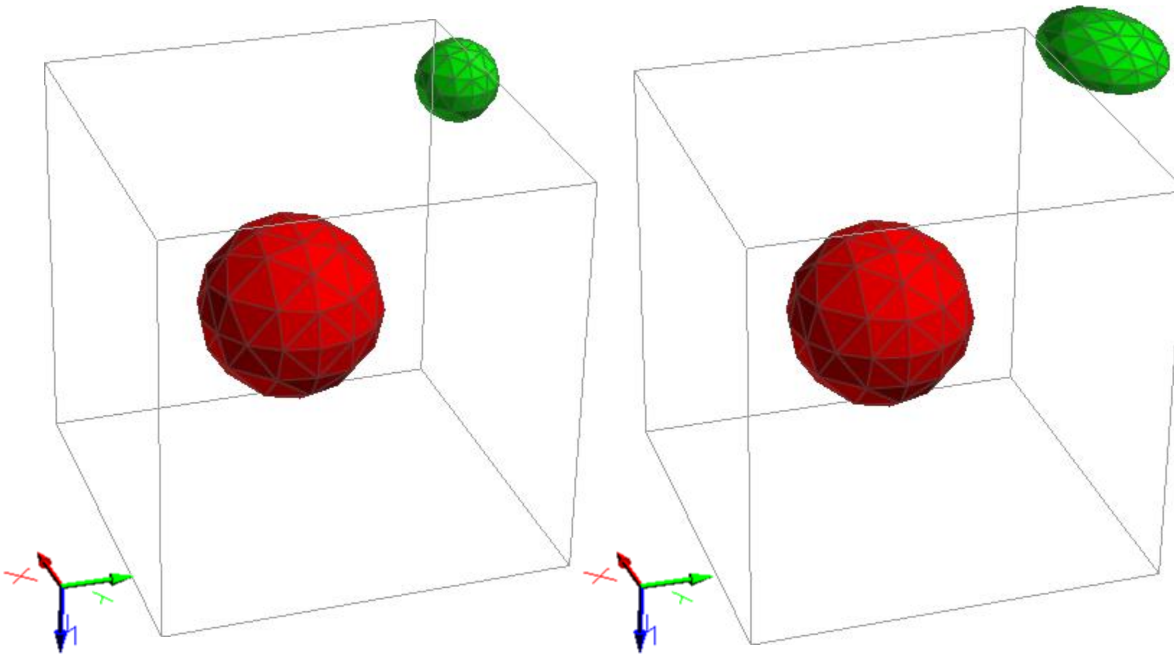
## Scale Mesh

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 scaled on the right box and define the scaling factors for the X, Y, and Z directions in **Scale X**, **Scale Y**, and **Scale Z**.




For the two-sphere example from the previous page, the comparison of the meshes before with scaling factors of 3, 2, 1 for X, Y, and Z are shown below.



If the mesh of the green particle is saved in ASCII format, it can be seen that the X, Y, and Z coordinates after scaling are 3, 2, and 1 times the original values, respectively. Please note that after scaling, also the center of the object is shifted by the same factor (See the green particle before and after scaling for reference).

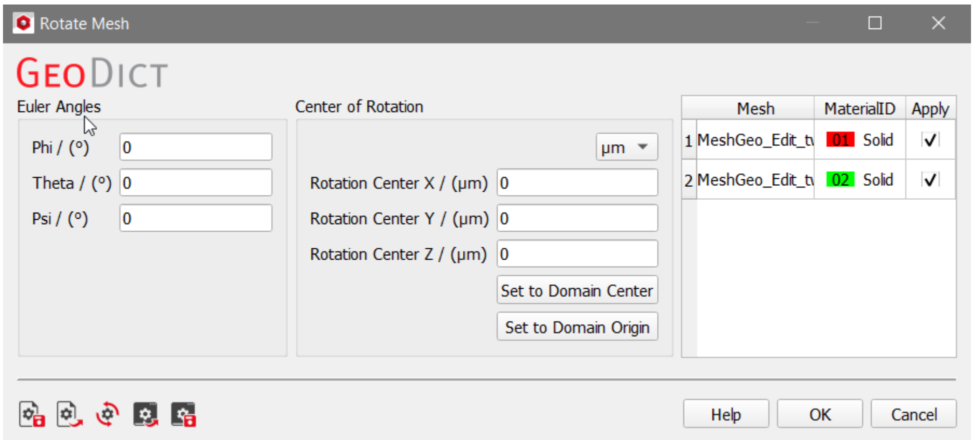
Here, below, the first image shows the first lines of values in STL before scaling. The second image shows the values after scaling.

```
solid 1
  facet normal +9.626375e-01 -1.914798e-01 +1.914798e-01
    outer loop
      vertex 5.461940e-02 4.808658e-02 5.000000e-02
      vertex 5.500000e-02 5.000000e-02 5.000000e-02
      vertex 5.461940e-02 5.000000e-02 5.191342e-02
    endloop
  endfacet
  facet normal +8.177519e-01 -1.808828e-01 +5.464093e-01
    outer loop
      vertex 5.408248e-02 4.795876e-02 5.204124e-02
      vertex 5.461940e-02 5.000000e-02 5.191342e-02
      vertex 5.353553e-02 5.000000e-02 5.353553e-02
    endloop
  endfacet
```

```
solid 1
  facet normal +8.318570e-01 -2.481991e-01 +4.963982e-01
    outer loop
      vertex 1.638582e-01 9.617316e-02 5.000000e-02
      vertex 1.650000e-01 1.000000e-01 5.000000e-02
      vertex 1.638582e-01 1.000000e-01 5.191342e-02
    endloop
  endfacet
  facet normal +4.415829e-01 -1.465140e-01 +8.851769e-01
    outer loop
      vertex 1.622474e-01 9.591752e-02 5.204124e-02
      vertex 1.638582e-01 1.000000e-01 5.191342e-02
      vertex 1.606066e-01 1.000000e-01 5.353553e-02
    endloop
  endfacet
```

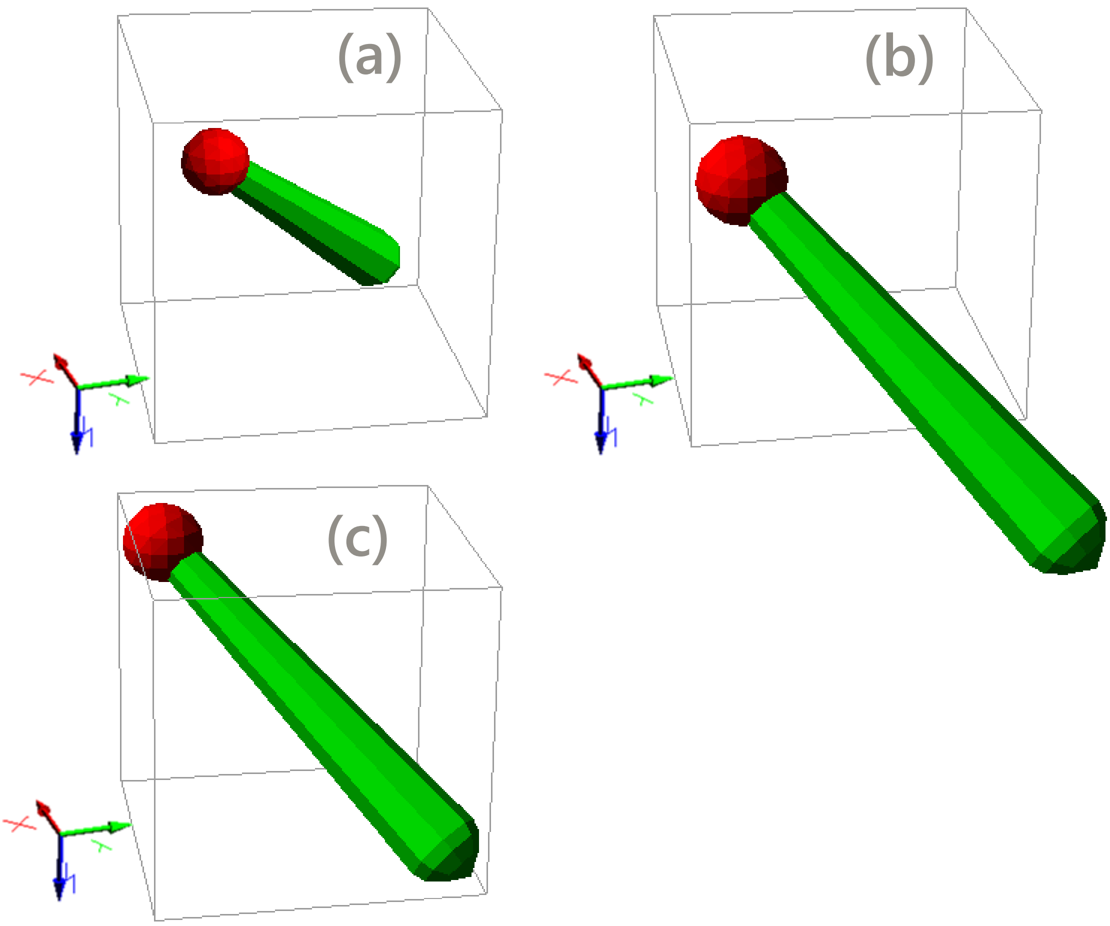
## Rotate Mesh

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



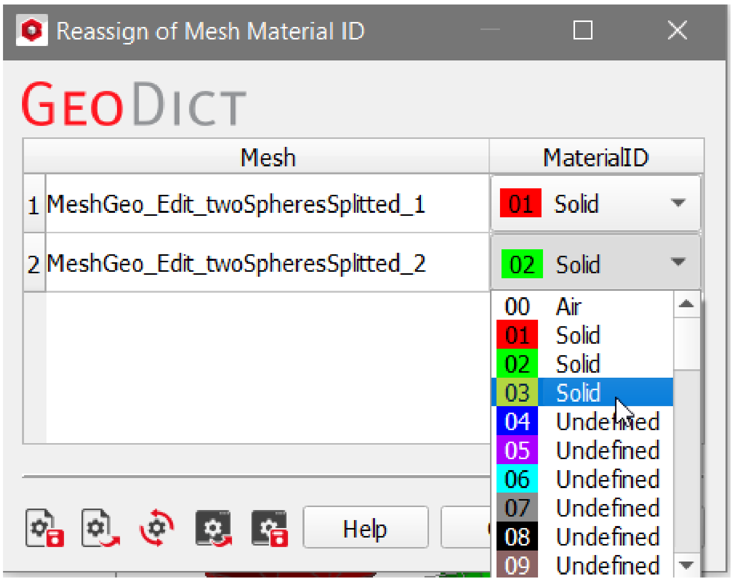

Enter the Euler Angles **Phi**, **Theta**, **Psi** by which the selected mesh should rotate. More details about Euler Angles can be found in GadGeo handbook. The rotation can be around either the domain center or domain origin.

An example of mesh rotation is shown below. Image (a) shows the original mesh of the match structure. Image (b) shows the mesh after the rotation of Phi with 90° around the domain origin, and Image (c) shows the rotation of Phi with 90° around the domain center.

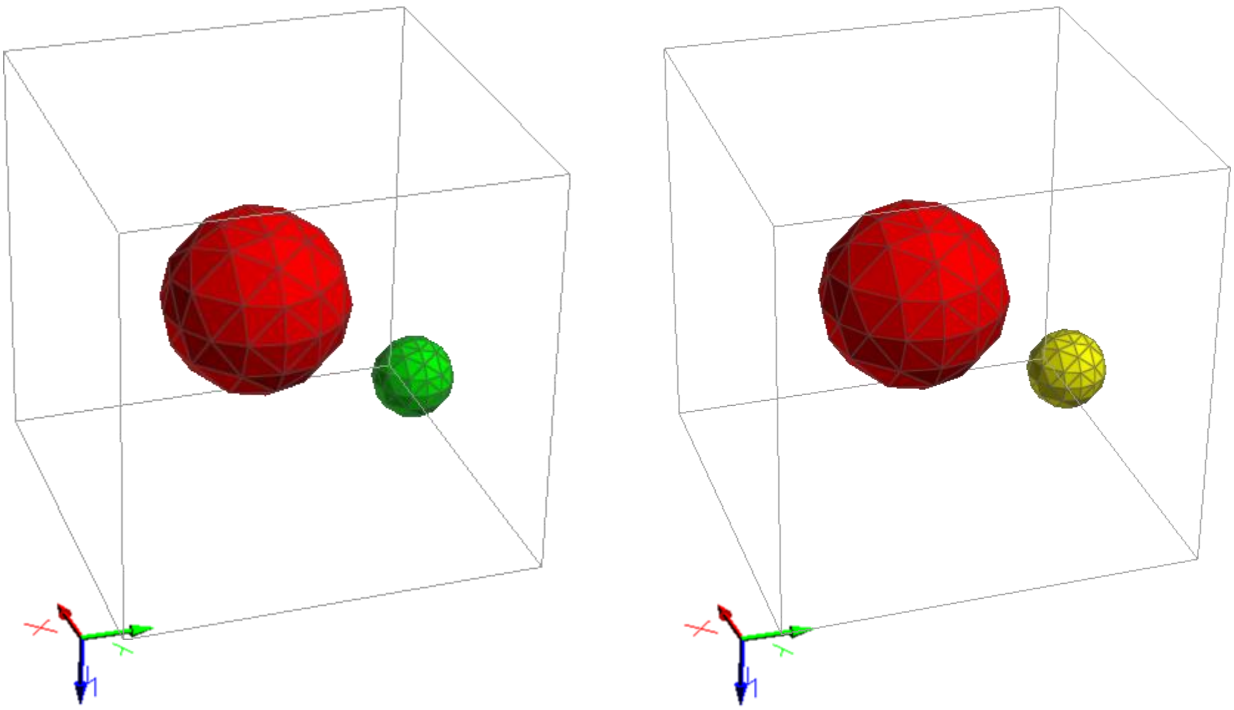


## Reassign Mesh Material ID

Each mesh can be assigned to a different material ID with **Reassign Mesh Material ID**. Click **Edit...** to open the dialog.



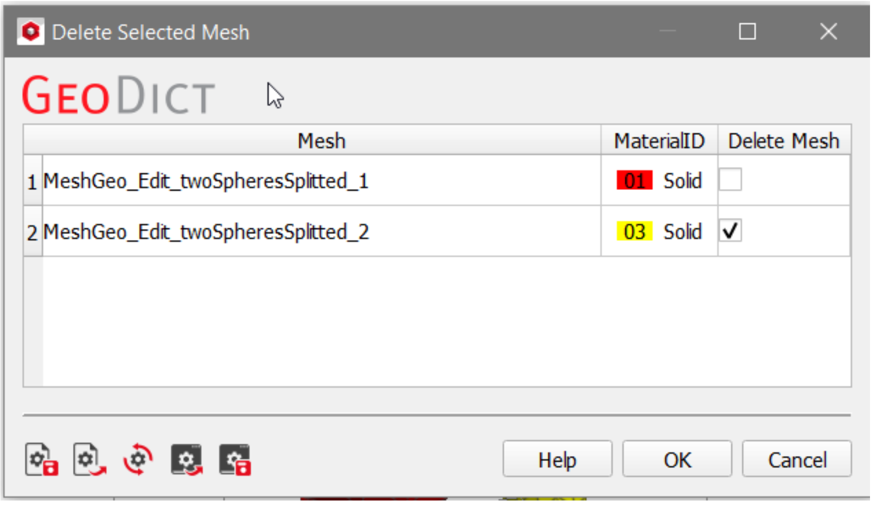

In the **Mesh** column, the currently loaded meshes are listed and the corresponding Material IDs are shown on the right. Each of the Material IDs can be reassigned. For the example below, the Material ID of the smaller sphere is originally 02, we change it to Material 03 and click **OK**. The Material ID for the smaller particle now becomes 03.



**Reassign Mesh Material ID** is often used after Split Meshes into Connected Components that multiple meshes with a same material ID are created for different components.

## Delete Mesh

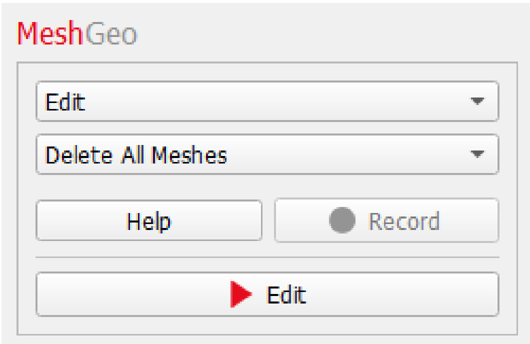
The selected meshes can be deleted with **Delete Mesh**. Click **Edit...** to open the **Delete Selected Mesh** dialog, the meshes currently loaded in memory are shown in the **Mesh** column.




Choose the meshes to be deleted by clicking the checkboxes in the **Delete Mesh** column, then click **OK** to go back to the MeshGeo section. Click **Edit** to remove the selected meshes.

## Delete All Meshes

**Delete All Meshes** deletes all the surface triangulations currently in memory. No options need to be set. Click I**Edit** after choosing **Delete All Meshes** to remove all the meshes.



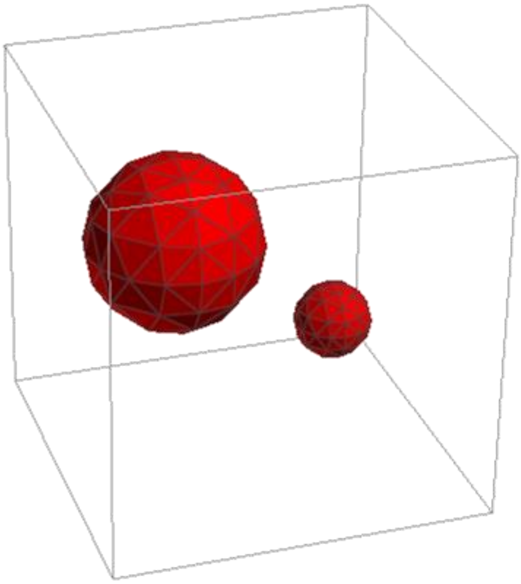

Alternatively, by clicking Menu **File** → **Discard Triangulation** all loaded meshes can also be deleted.

## Split Meshes into Connected Components

With **Split Meshes into Connected Components**, an individual mesh 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.

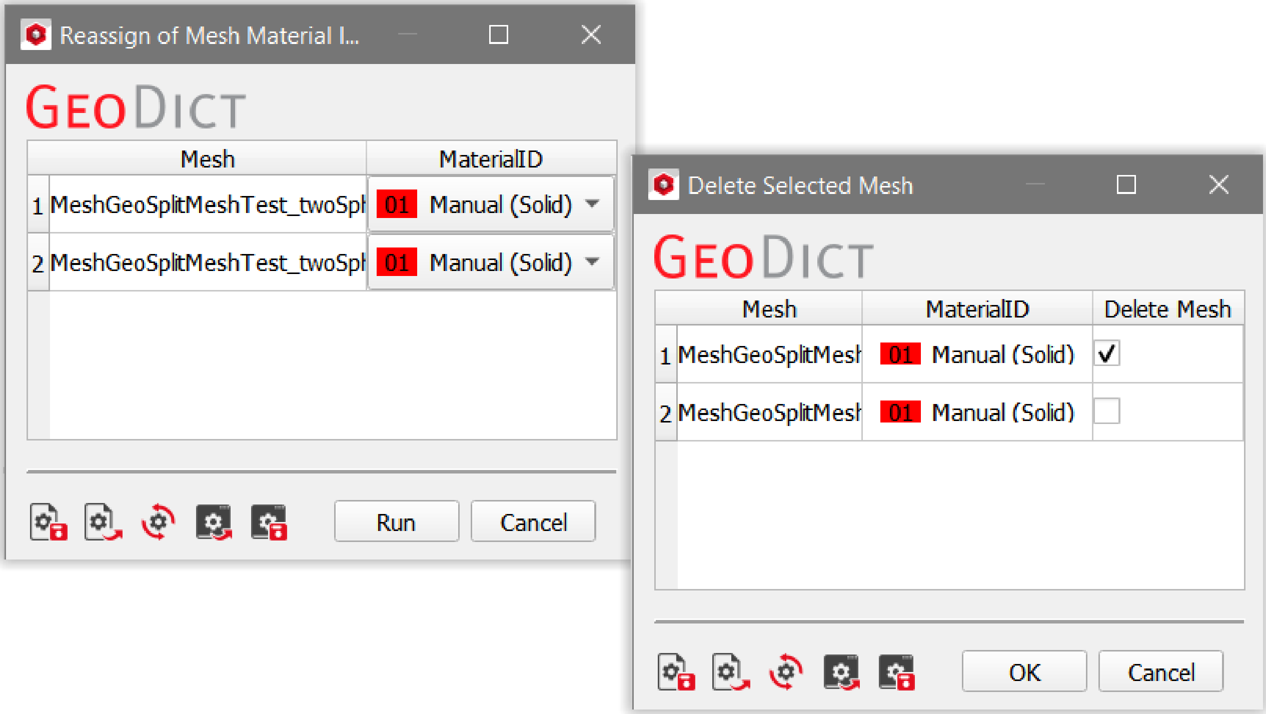
Open the settings dialog for **Reassign Mesh Material ID** (see below in page 33) and observe that only one mesh is shown.




Now go back to **Split Meshes into Connected Components**. Since the command detects the connected components automatically and separates them into individual meshes, no settings are necessary. Just click **Edit** to run the command.

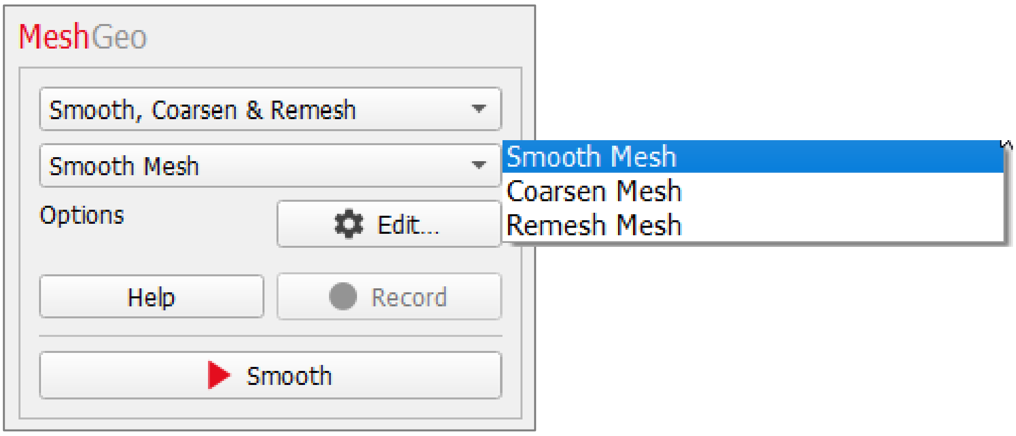
When opening the dialog of **Reassign Mesh Material ID** again, two mesh handles now appear for the two spheres.

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



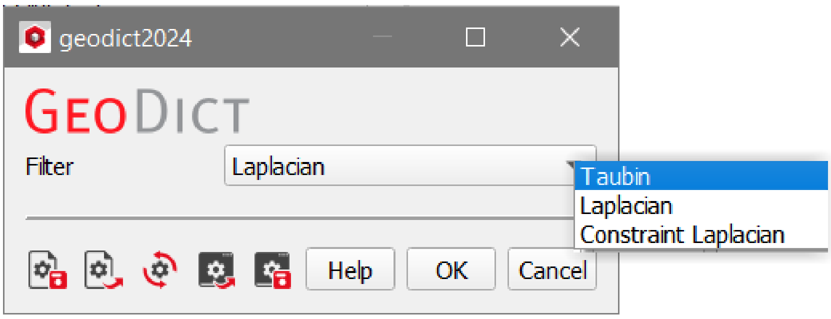
# Smooth, Coarsen & Remesh

The currently loaded triangulation can be smoothened, coarsened, as well as re-meshed by choosing the corresponding options under **Smooth, Coarsen & Remesh**.



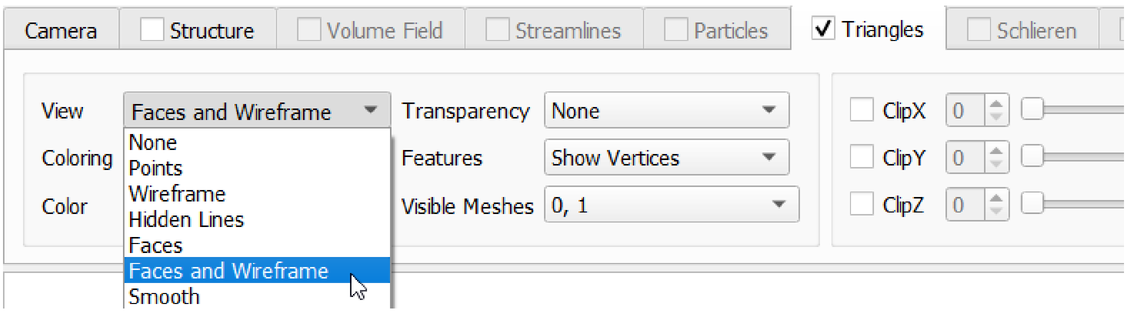

## Smooth Mesh

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.

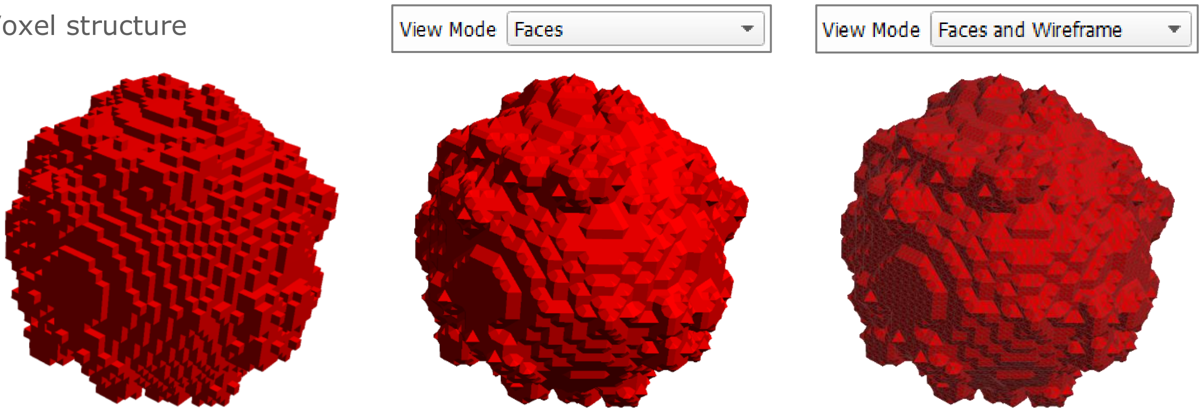



Smoothing is best shown on a structure with an irregular surface. The example structure created here is a grain with surface roughness. The original mesh before smoothing is created with **Create Mesh** → **Create Voxel Mesh** and the Multi Material mode (see page 9).

To observe the differences between the 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).

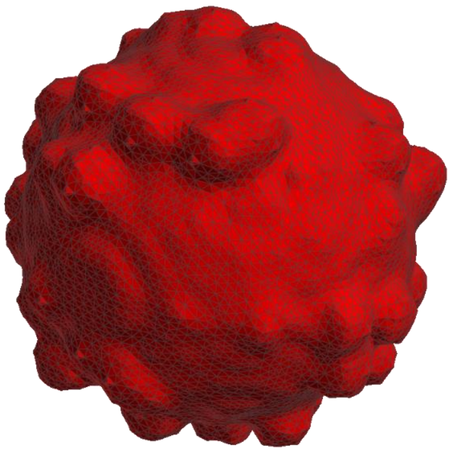


Voxel structure

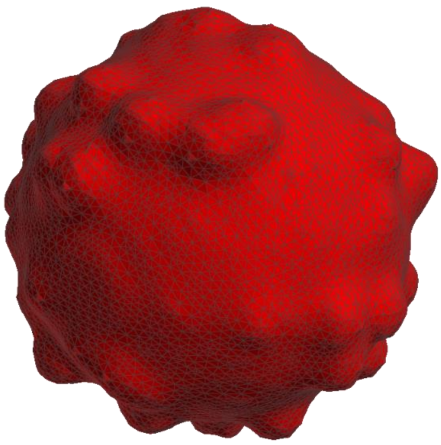



The **Taubin** filter is the default smoothing filter in MeshGeo. This filter iteratively combines different Laplacian filters to provide better volume preservation. The runtime for **Taubin** is longer than for Laplacian. In the screenshot on the right, observe that the mesh is rougher than the result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

More information on the Taubin filter in (1).

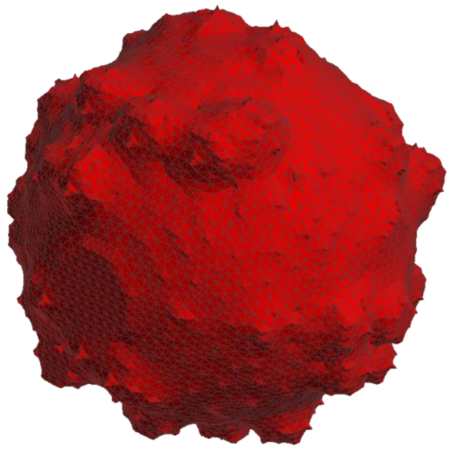


The **Laplacian** filter is a simple and fast option for mesh smoothing. However, this filter causes mesh shrinking for most geometries. For further information, see Wikipedia: Laplacian Smoothing.



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.

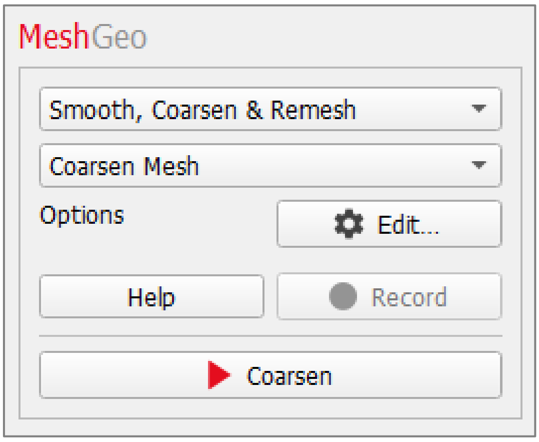


## COARSEN MESH

**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.

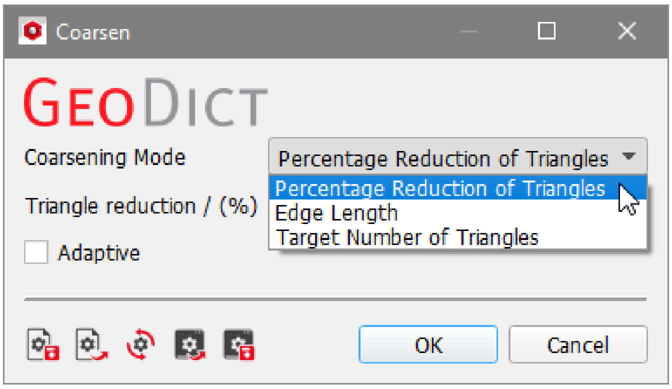
Generally, it is not supported to use Coarsening on the voxel surface mesh (**Create Mesh → Create Voxel Mesh → Voxel Surface**). Use the other voxel mesh modes, or smooth the voxel mesh before coarsening.




### 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**.

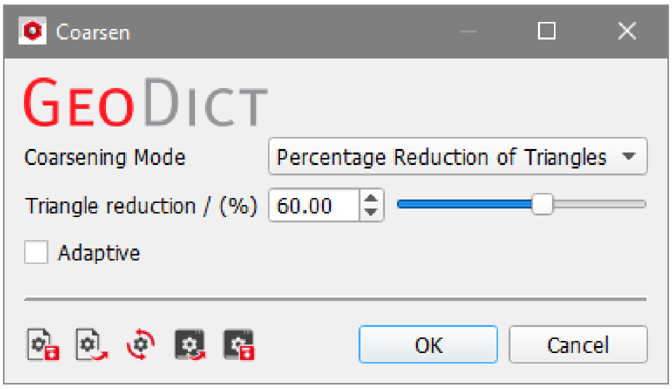
The coarsening algorithm reduces the number of triangles in the currently loaded mesh by combining short edges to longer edges.



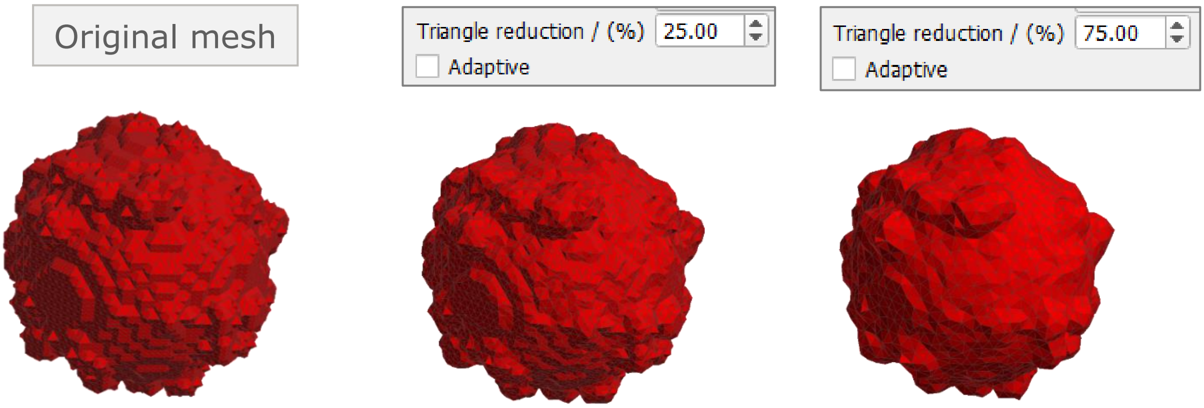

#### PERCENTAGE REDUCTION OF TRIANGLES

**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.

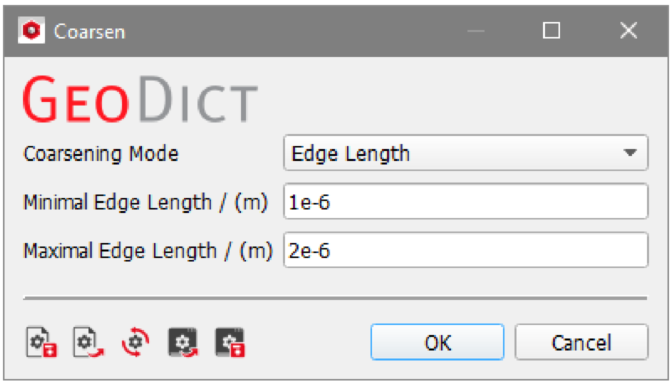



## Edge Length

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.

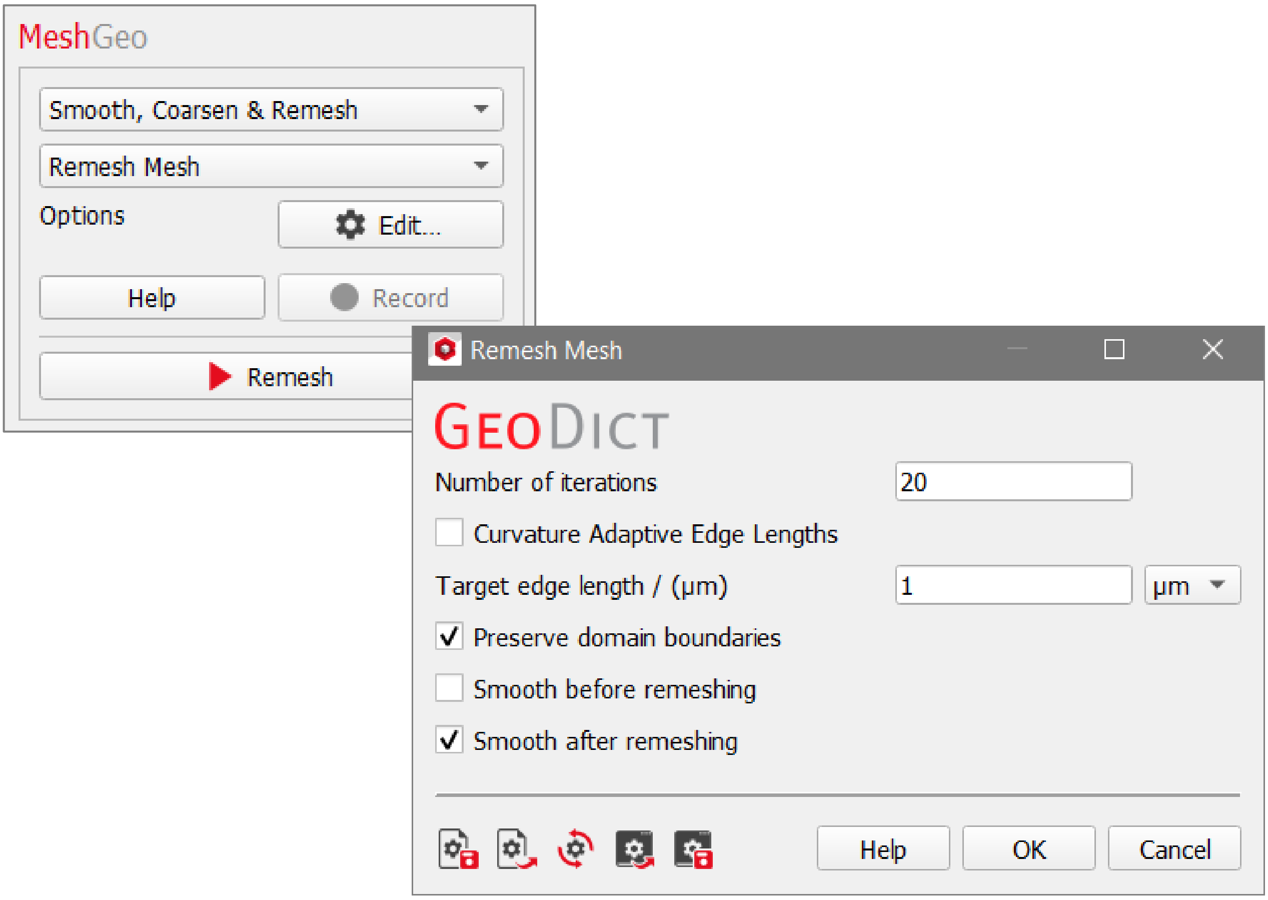
## ADAPTIVE

**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.

## Remesh Mesh

The loaded mesh can be re-meshed with a given edge length for the 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.

### Curvature Adaptive Edge Lengths

When the option **Curvature Adaptive Edge Lengths** is activated, the algorithm performs a remeshing with an adaptive triangle size based on local curvature values instead of a fixed target size.

### 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.

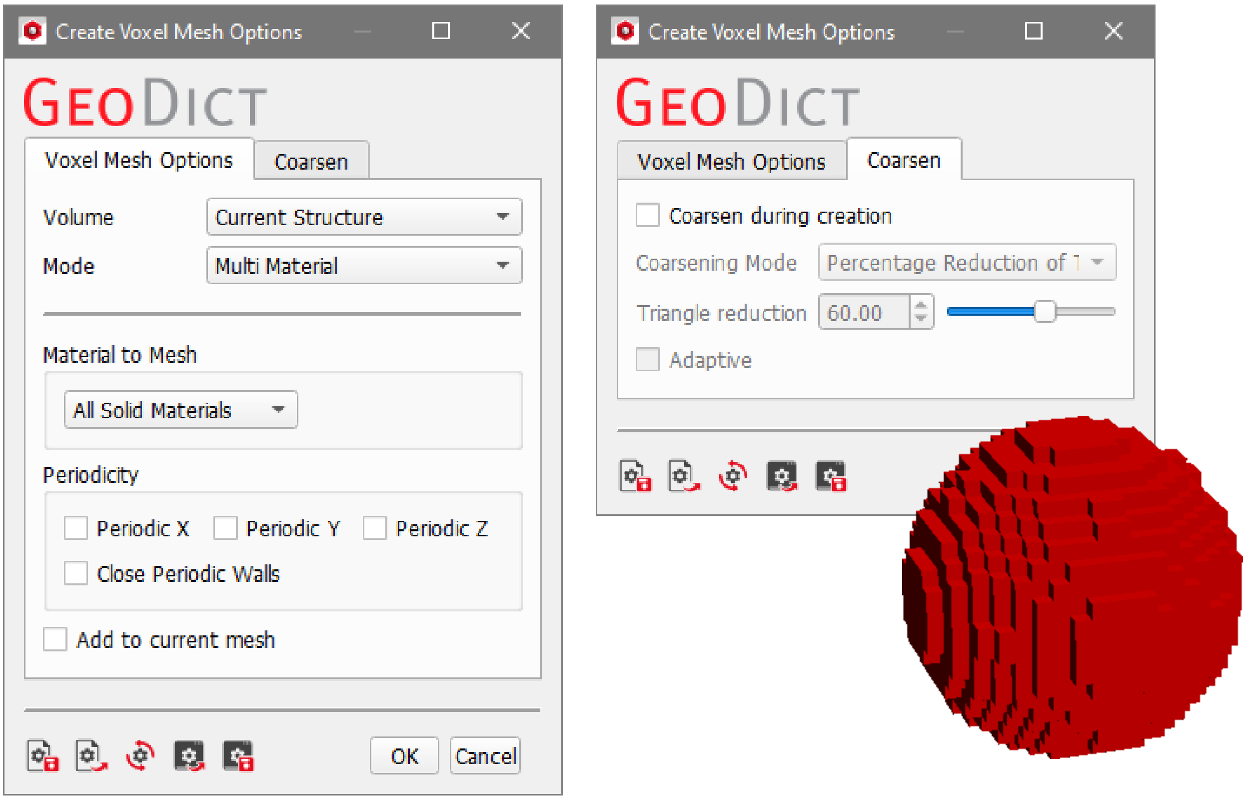
## Smooth before remeshing

When **Smooth before remeshing** is chosen, the smoothing step is performed before the remeshing algorithm starts. This is recommended when the **Curvature Adaptive Edge Lengths** is used.

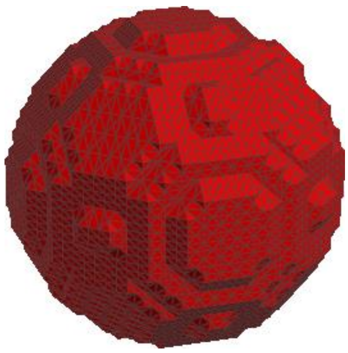
## Smooth after remeshing

When **Smooth after remeshing** is chosen, the smoothing step is preformed after the remeshing algorithm finishes. It often improves the quality of the resulting mesh. However, it may also result in features such as sharp edges not being preserved if the mesh resolution is not high enough. Technically, they are also not preserved during the remeshing, but the additional smoothing step at the end will erode those even further.

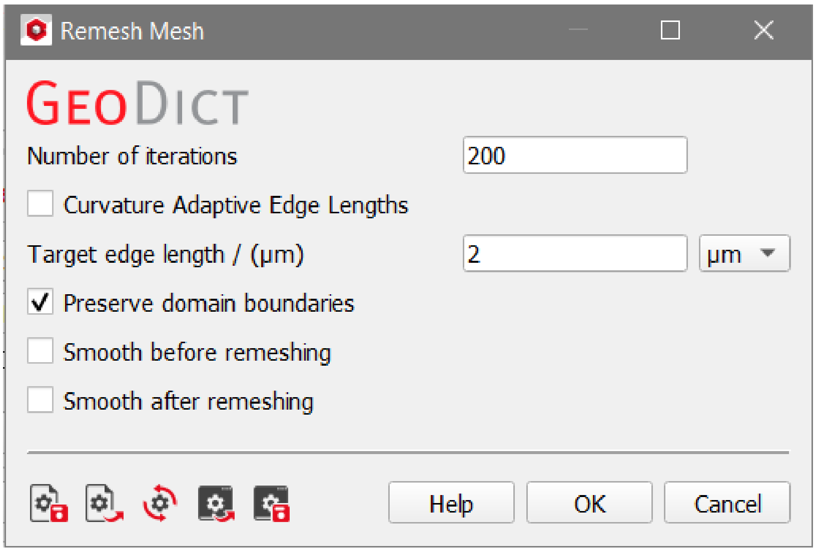
To illustrate the features, an example of a ball with 20 µm diameter with the voxel length 1 µm is used for meshing (**Create Mesh**), remeshing (**Remesh Mesh**), and smoothing (**Smooth Mesh**) to show the different meshes created.

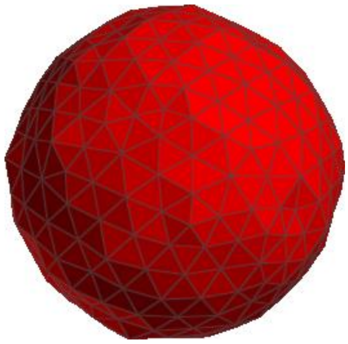



First, a voxel mesh is created with the settings shown in the dialogs above, without any **Coarsen**.

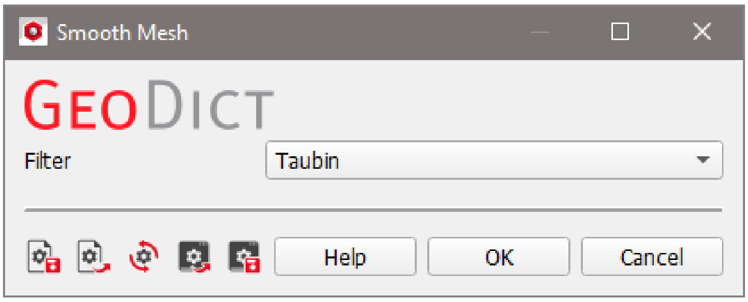


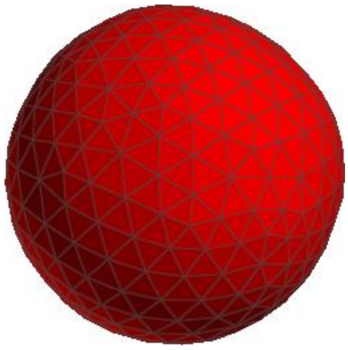
**Remesh Mesh** is executed with 200 as the **Number of iterations** and 2 µm as the **Target edge length**.



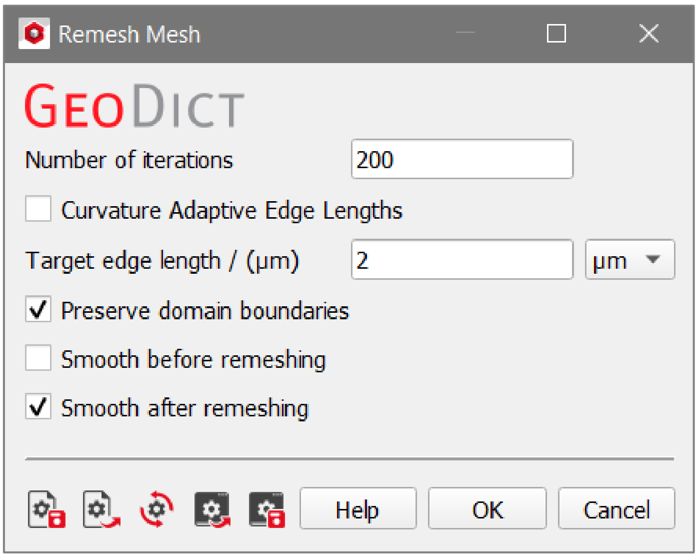



The mesh is then smoothed with the **Taubin** filter



The two steps of **Remesh Mesh** and **Smooth Mesh** can be combined into one step using **Smooth after remeshing** in **Remesh Mesh**. Then, only the mesh after the final step is shown.

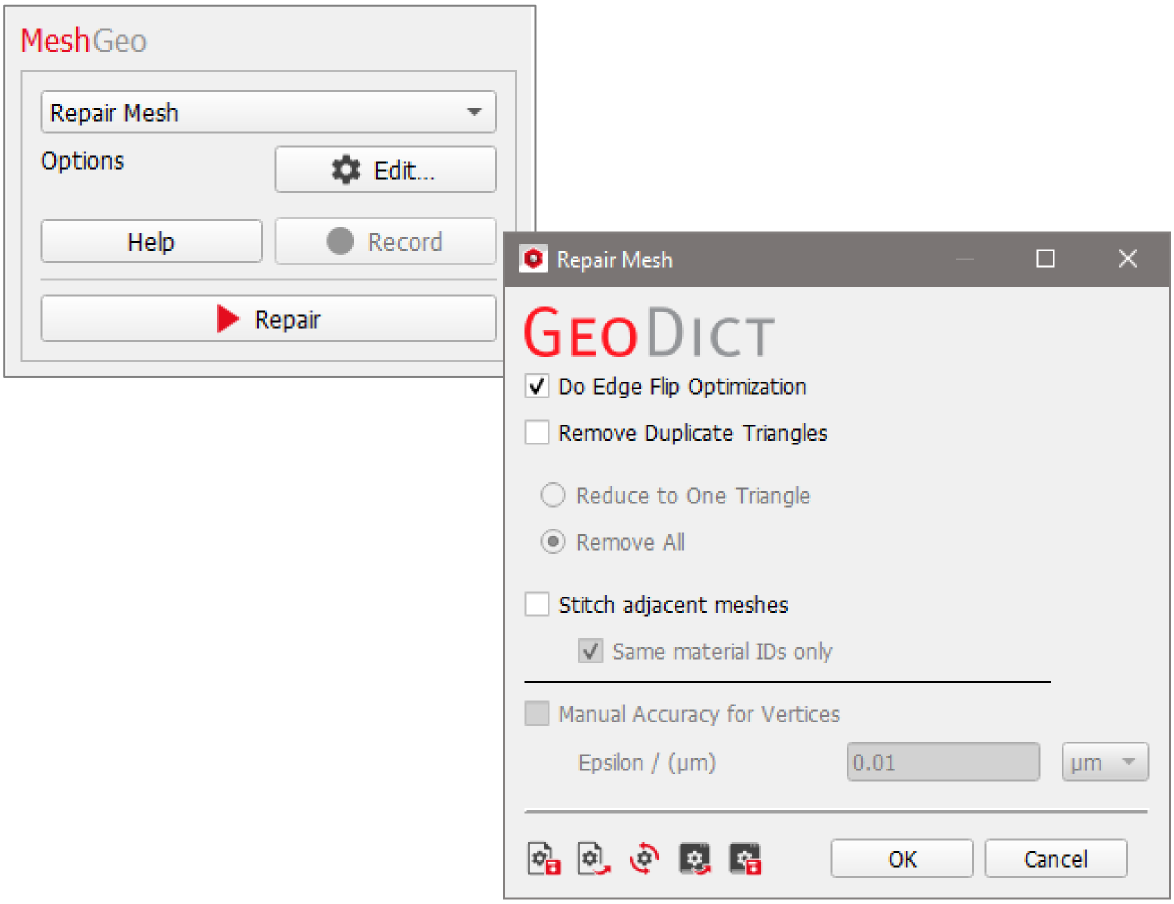


# Repair Mesh

**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.

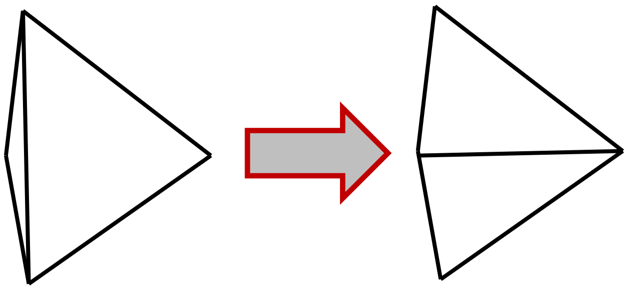



When all settings have been chosen, click **Repair** at the bottom of the **MeshGeo** section to create the mesh.

## 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°, 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 options 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

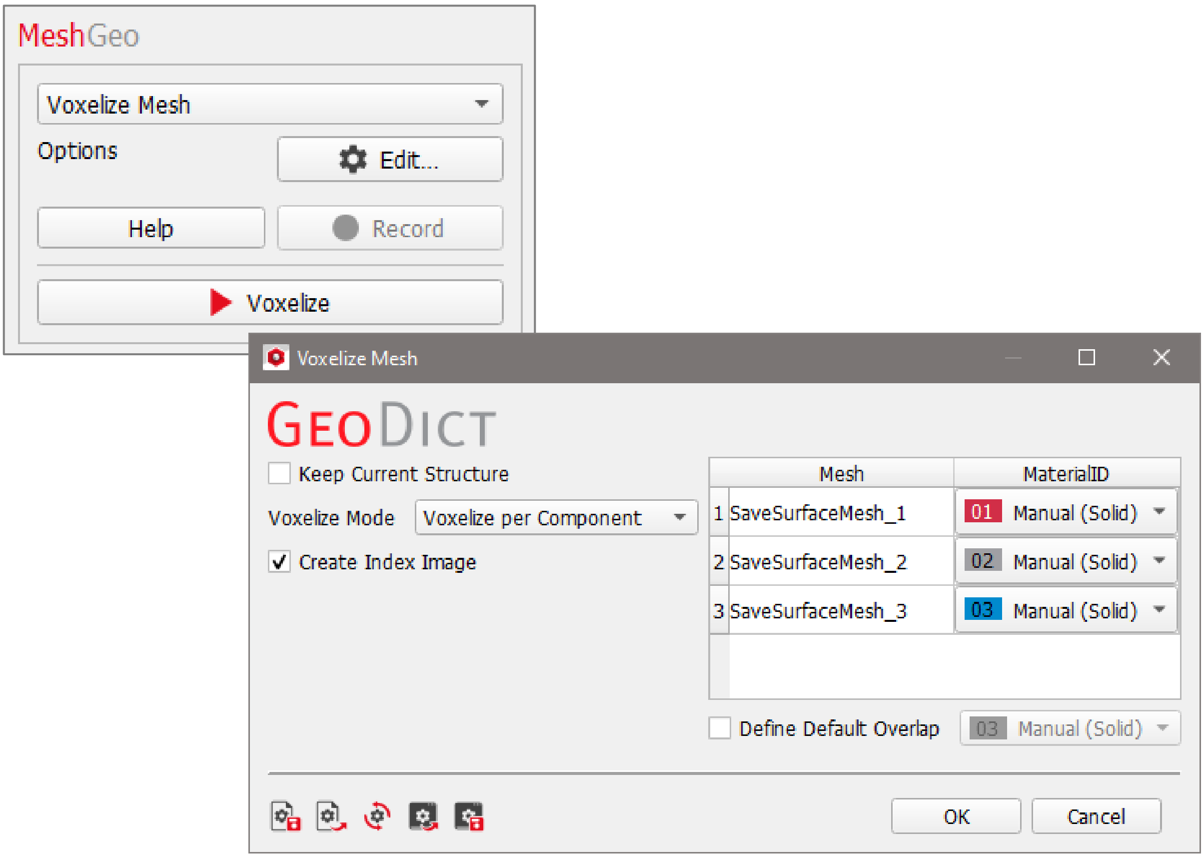
The option of **Manual Accuracy for Vertices** is valid only when **Remove Duplicate Triangles** is used to define a threshold to compensate for numerical inaccuracies in vertex coordinates. The mesh vertices with at most the distance given for **Epsilon / (µm)** is treated as equal.

# 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.

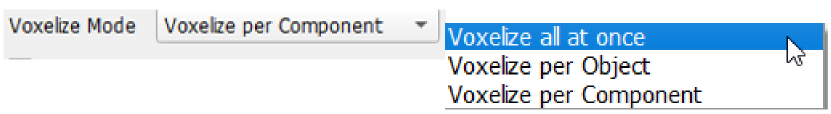



Some of the settings in the **Voxelize Mesh** dialog are analogous to the ImportGeo-CAD dialog. The more detailed explanation of the settings can be found in the [ImportGeo-CAD](ImportGeo-CAD) handbook.

## Keep Current Structure

If **Keep Current Structure** is chosen, the currently loaded structure is not removed and the new structure voxelized from the mesh will be added to the current structure.

## Voxelize Mode



There are three voxelize modes: **Voxelize all at once**, **Voxelize per Object**, and **Voxelize per Component**.

**Voxelize per Object** voxelizes the mesh using object indices saved in the mesh, and **Voxelize per Component** creates the voxel geometry per connected component saved in the mesh. The different objects or components are converted to different Material IDs. The two options are also useful to handle overlapping objects. **Voxelize all at once** then does it all in one but differentiating neither the objects nor the components.

## Create Index Image

The option **Create Index Image** is only available for **Voxelize per Object** and **Voxelize per Component**. It creates an index image containing the object ID for **Voxelize per Object** or component ID for **Voxelize per Component**.

## Material ID selection

The table lists on the first column the currently loaded meshes and on the second column the Material ID that can be assigned to the corresponding meshes after voxelizing.

## Define Default Overlap

With this option, the user can assign specific material to overlap regions. If no default overlap is defined, the material ID of the overlap regions is automatically computed through a binary addition of the IDs of the overlap materials.

## 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/10.30423/userguide.geodict

Technical documentation:

**Sebastian Rief**

**Liping Cheng**

**Andreas Grießer**

**Barbara Planas**

Math2Market GmbH

Richard-Wagner-Str. 1, 67655 Kaiserslautern, Germany

www.geodict.com