MeshGeo

User Guide

GEODICT Release 2026

Published: June 2026

GEODICT

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

MeshGeo

1. MeshGeo 3

1.1 Create Mesh 5

1.1.1 Create Voxel Mesh 5

1.1.2 Create Object Mesh 15

1.2 Analyze 22

1.2.1 Compute Mesh Statistics 22

1.2.2 Compute Three-Phase Contact Line 24

1.3 Edit 27

1.3.1 Shift Mesh 27

1.3.2 Scale Mesh 28

1.3.3 Rotate Mesh 32

1.3.4 Reassign Mesh Material ID 33

1.3.5 Delete Selected Mesh 35

1.3.6 Delete All Meshes 35

1.3.7 Split Meshes into Connected Components 36

1.4 Smooth, Coarsen & Remesh 38

1.4.1 Smooth Mesh 38

1.4.2 Coarsen Mesh 39

1.4.3 Remesh Mesh 42

1.5 Repair Mesh 47

1.6 Voxelize Mesh 49

1.7 Save Mesh 51

1.8 Export as Mesh 54

1.9 References 55

1 MeshGeo

Use MeshGeo to convert GeoDict data formats to a smooth surface mesh with sub-voxel precision, analyze and edit the mesh, and save it to your hard disk for further use with external software.

Meshes can be generated directly from

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

MeshGeo keeps 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**.

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

New in GeoDict 2026

- The Save Mesh 51 function for IGES and STEP formats has been updated to remove the CADlook dependency.
- New Export Object CAD command to export the analytic objects or their inverse pore space geometry directly in a CAD format.
- **Origin of Scaling** can be chosen for Scale Mesh 28.

Learn to use MeshGeo:

- Create Mesh 5
  Create surface meshes for voxel structures and analytic objects.
- Analyze 22
  Evaluate the mesh quality by finding mesh statistics and and the interfaces between different materials.
- Edit 27
  Modify the loaded meshes.
- Smooth, Coarsen & Remesh 38
  Smooth, coarsen, and remesh the loaded triangulation.
- Repair Mesh 47
  Repair the loaded mesh.
- Voxelize Mesh 49
  Generate a voxel structure based on the currently loaded meshes.

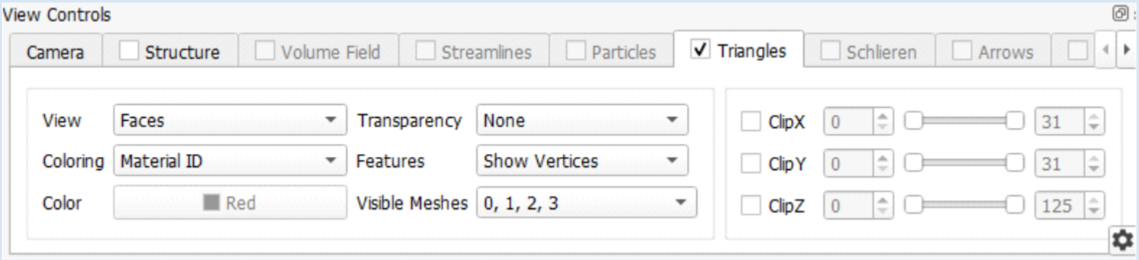
Learn to use MeshGeo:

- [Save Mesh] 51
  Write the currently loaded mesh to a file.
- [Export as Mesh] 54
  Create and save meshes directly into a file without leaving the triangles in memory.
- [References] 55
  Publications cited in this chapter.

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 User Guide.



Join the conversation

Visit the Community on [GeoDict Forum] to be inspired and get answers to top questions.
Find illustrative application examples on the [website].
Our tutorials and seminar videos give you a first hands-on experience. They are collected in the [Learning Center].
Send your [Support Request] to our technical Support.

1.1 Create Mesh

Select **Create Mesh** in MeshGeo 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 with **Voxel Mesh**. This voxel information can be a structure, a gray value image, an index image, or a volume field.

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

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

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

Commands of Create Mesh:

- Create Voxel Mesh 5
  Create triangulations from the available voxel information.
- Create Object Mesh 15
  Create triangulations from the analytic object data.

1.1.1 Create Voxel Mesh

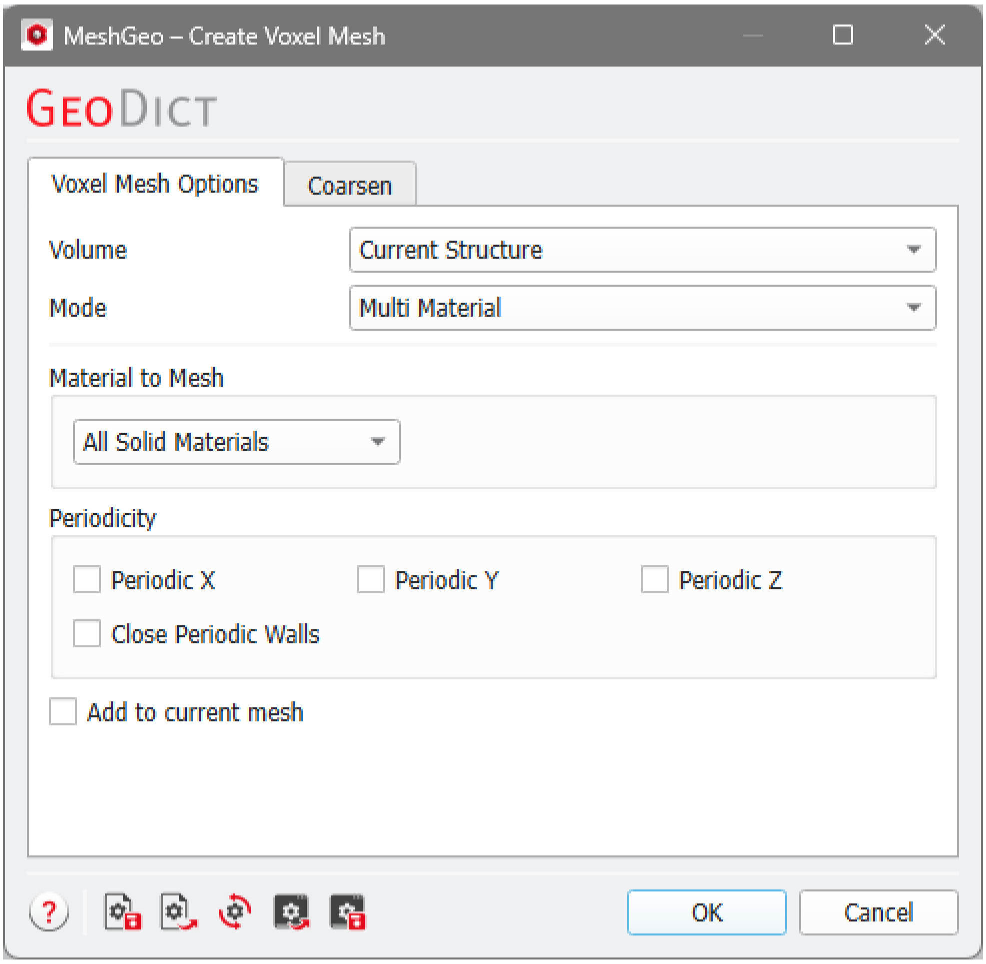
Create Voxel Mesh converts a voxel model into surface triangulations. This includes GeoDict voxel structures, loaded gray-value images, index images and volume fields. In the **Create Voxel Mesh** dialog, two tabs are available: **Voxel Mesh Options** and **Coarsen**.

The Create Voxel Mesh command:

- Voxel Mesh Options 5
  Set the options for the creation of the mesh.
- Coarsen 12
  Options for mesh coarsening.

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

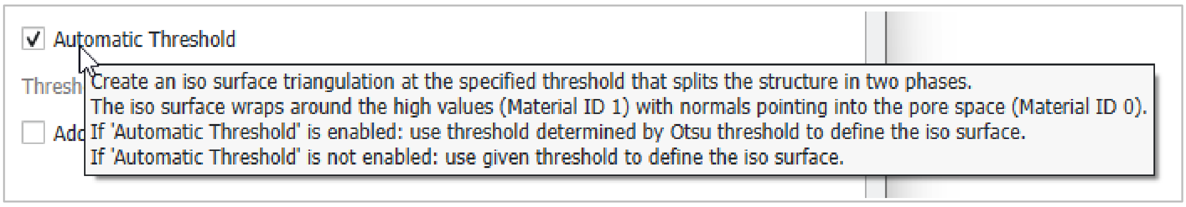
Current Structure

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

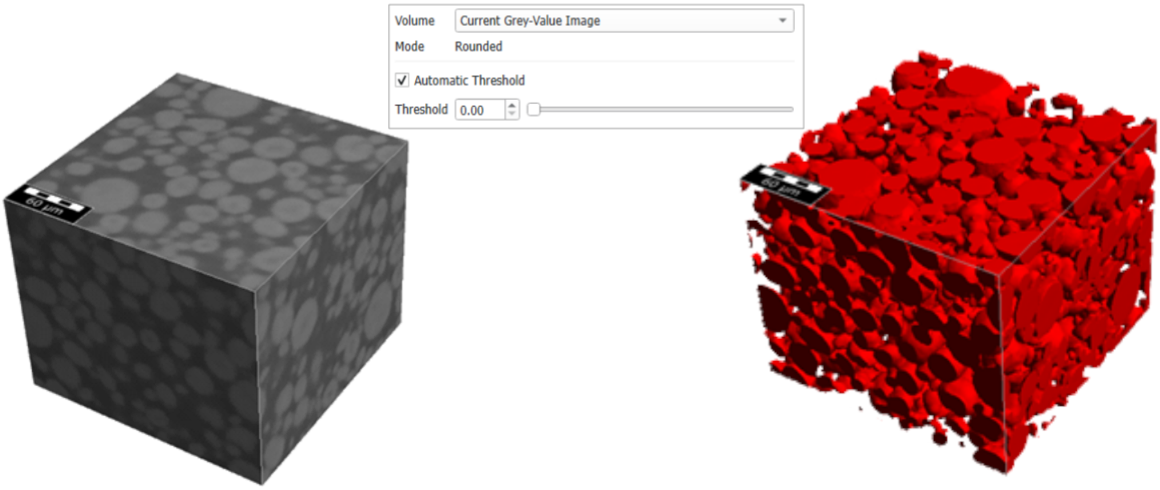
Current Grey-Value Image

Instead of first converting a gray-value image to a voxel structure (with ImportGeo-Vol), it is possible to directly mesh the **Current Grey-Value Image**. In this case, the **Mode** is fixed to **Rounded** 9. Based on the given **Threshold**, an isosurface is created that splits the structure in two phases. The normal of the isosurface points into the pore space. If **Automatic Threshold** is selected, the Otsu algorithm is used to determine the threshold. The tooltip also displays the explanation when the mouse hovers over **Automatic**

Threshold.



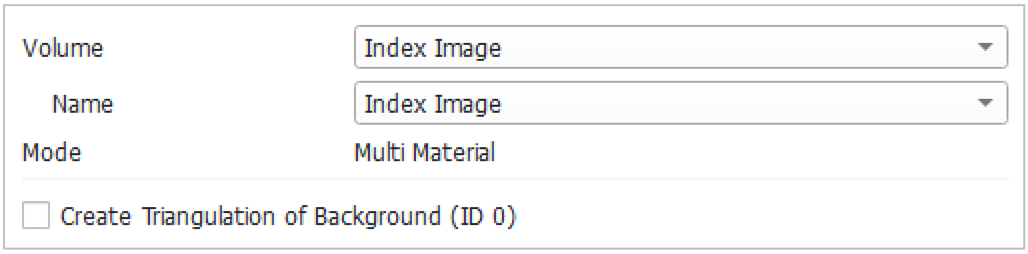
With this option, all information from the gray-value image can be used. The created mesh is smoother than a mesh created from a segmented voxel structure, since the gray-value information allows to obtain sub-voxel precision.



Index Image

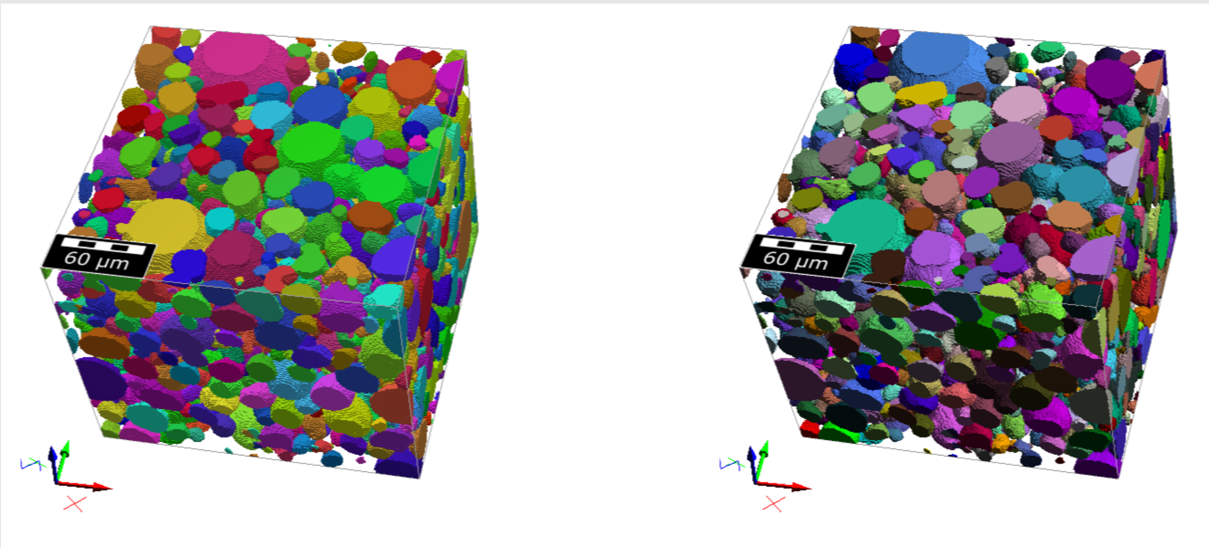
By choosing an **Index Image** as Volume, it is possible to use the object indices to create **Multi Material** triangulations. Analogously, the object indices from GrainFind or FiberFind results can be used in the same way.

Each object with a given index is converted to its own mesh, and the meshes for different objects do not overlap. The **Mode** is fixed to **Multi Material** 9, since this is the option that keeps the object information and preserves the partition of unity.



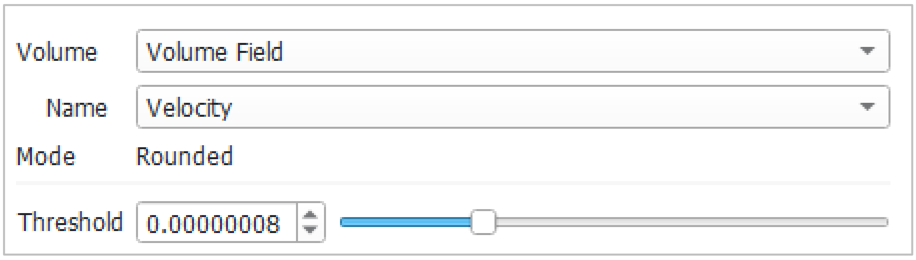
If **Create Triangulation of Background (ID 0)** is checked, the triangulation is created also for the Index 0 in the image, usually pore material.

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

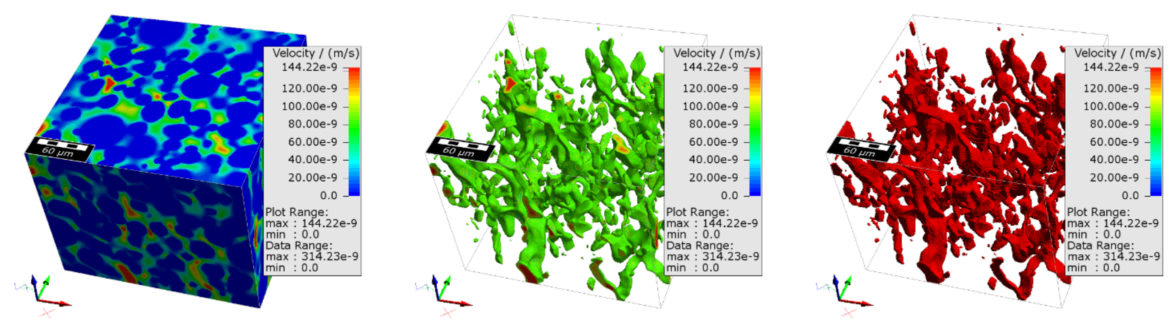


Volume Field

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



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



Mode

Four **Mode**s can be used to export voxel data: **Voxel Surface**, **Rounded**, **Smooth**, or **Multi**

Material. **Multi Material** is the default. Which options are available depends on the selected option for **Volume**: For example, if the **Current Grey-Value Image** is selected, only **Rounded** is available.

Voxel Surface

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

Rounded

Rounded works analogously to the Smooth export, but it focuses on triangle quality instead of smoothness. The exported surface is constrained to the domain and the surfaces of multiple materials may intersect. This export mode is most suitable for numerical simulations with only one material phase (e.g., flow simulations in the pore space).

Smooth

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

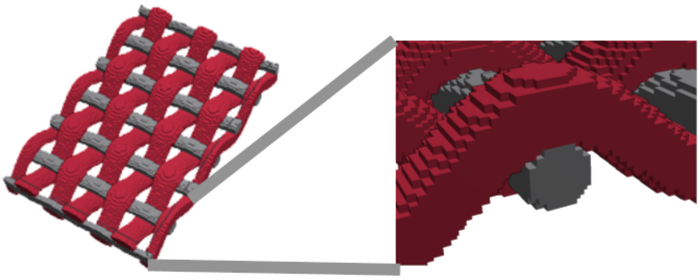
Multi Material

With the **Multi Material** option, the meshes for the different materials do not overlap. They have perfect contact at the points where different materials touch. When all materials are exported, a partition of unity is created. This means that each point in the domain is enclosed in exactly one surface. This property is preserved with further processing steps (e.g., **Smooth Mesh** 38 or **Coarsen Mesh** 39) in MeshGeo.

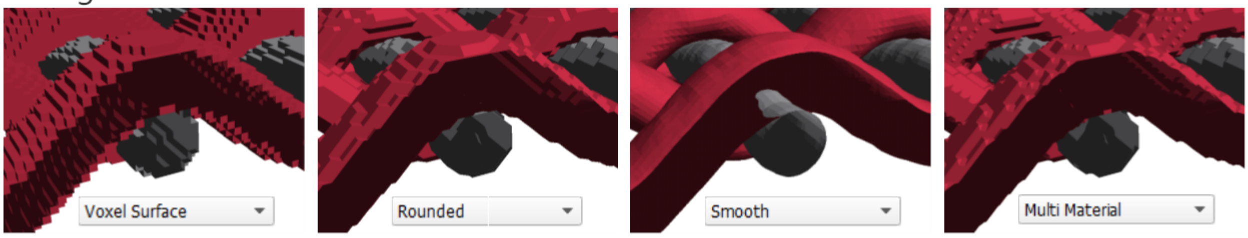
Multi Material meshes are constrained to the domain. This export mode is suitable to mesh multi-material microstructures for numerical simulations.

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

Voxel Structure in GeoDict

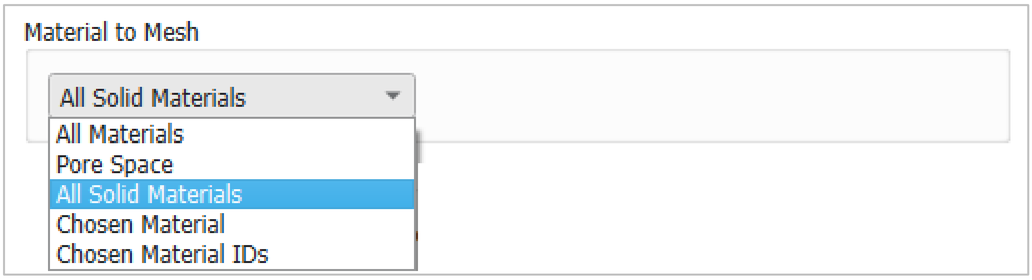


Triangulations in different modes

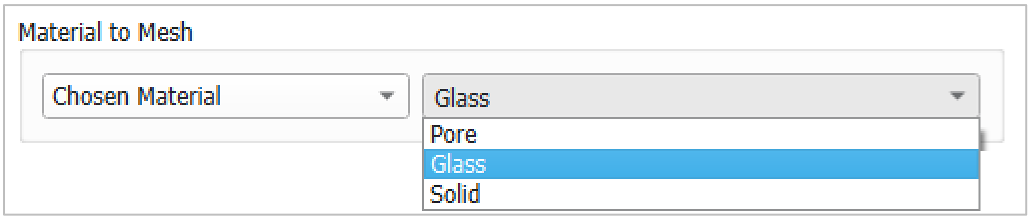


Material to Mesh

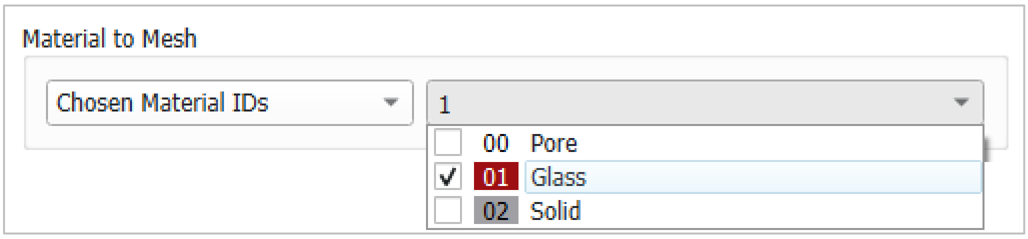
Select the Material IDs to be meshed with the options under **Material to Mesh**. With the first three options, either **All Materials**, the **Pore Space**, or **All Solid Materials** are meshed. **All Solid Materials** is the default, since this is the best choice for most use cases.



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

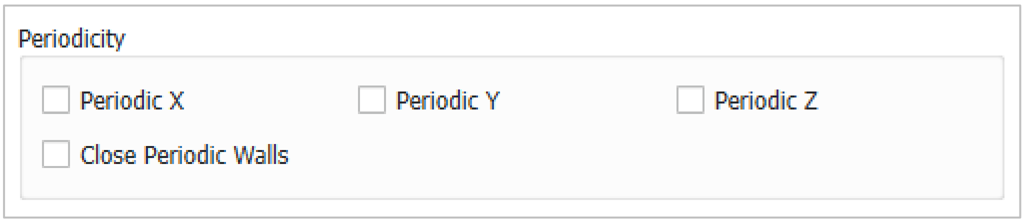


With **Chosen Material IDs**, individual Material IDs can be selected, which can have assigned different materials.



Periodicity

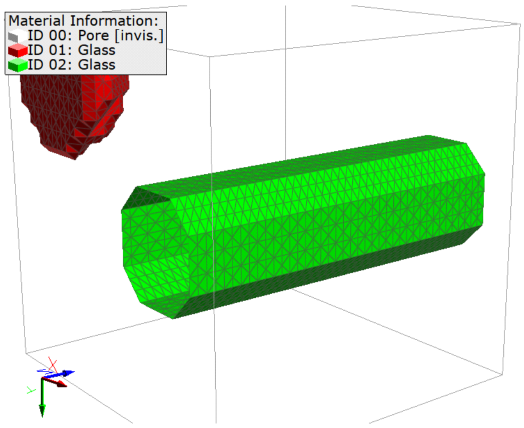
The boundary condition defines how the triangulation algorithm deals with boundary voxels.



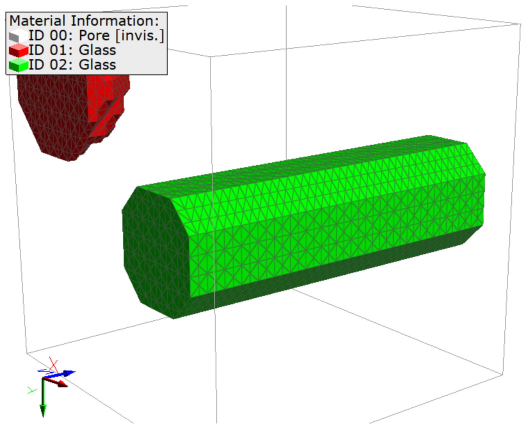
When **Periodicity** is selected, the mesh of one end is periodic to the other end of the structure in the chosen direction. This option should only be used if the original structure is periodic.

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

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



Periodic

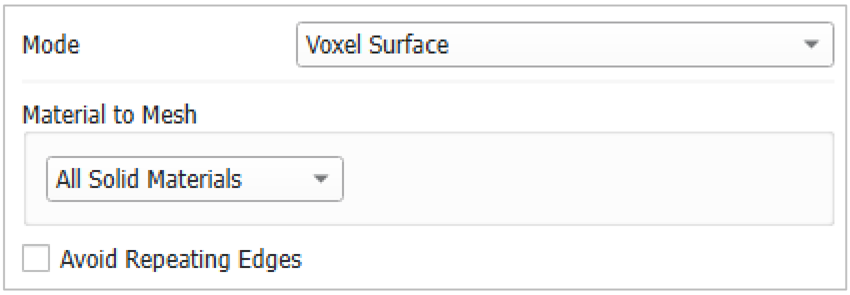


not Periodic

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

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

Avoid Repeating Edges

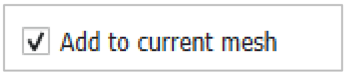


This option is available when **Voxel Surface** mode is chosen. Where different materials are in contact, their voxel surfaces share the same edges. This is not allowed in a mesh in the STL format. With **Avoid Repeating Edges**, these edges are shifted slightly inwards the material. Then, each material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

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

Add to current mesh

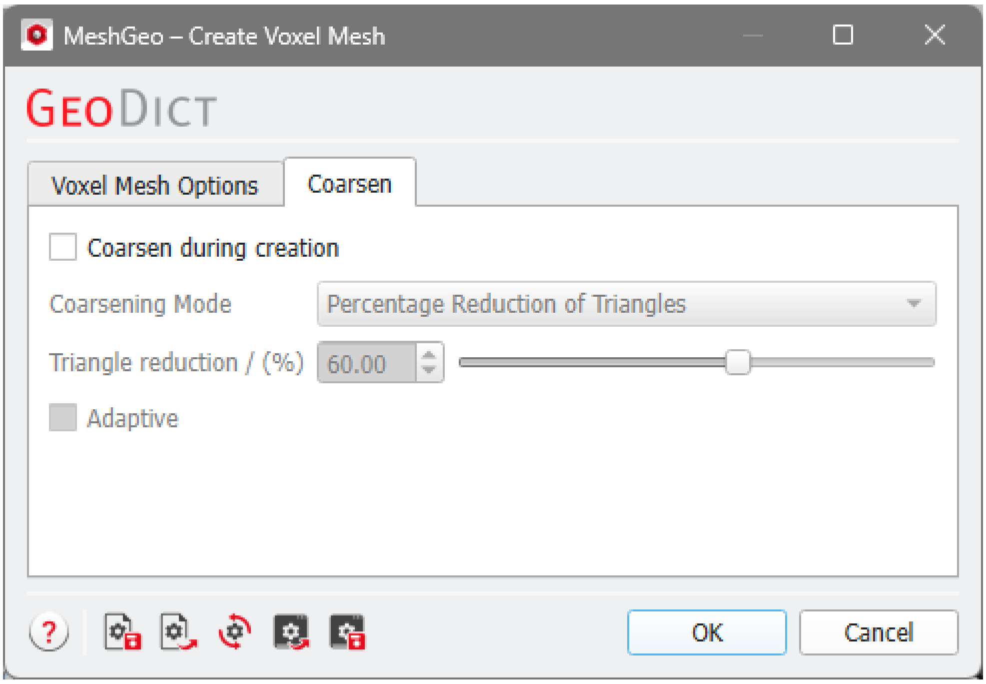
When this option is selected and a mesh is already in memory, the new mesh is added to that mesh. Multiple meshes can then be edited together and saved to a single file.



Coarsen

Coarsening can be applied either during mesh creation or after it has been generated. The settings are the same, but the results may differ – especially if the **Adaptive** mode is used.

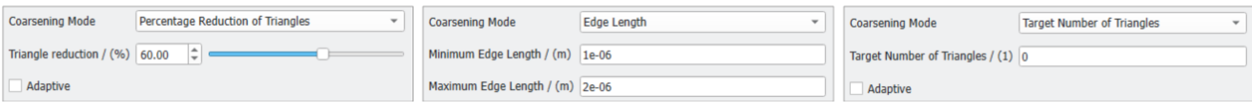
Check **Coarsen during creation** to enable coarsening during mesh creation. To coarsen the mesh after the creation use Coarsen Mesh 39.



Coarsening Mode

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

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

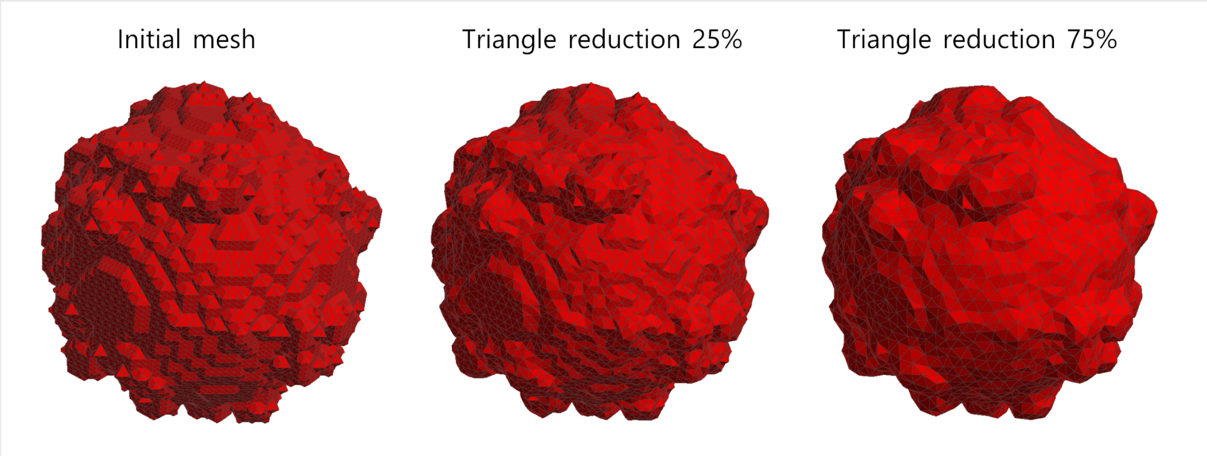


Percentage Reduction of Triangles

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

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

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



Edge Length

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

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

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

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

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

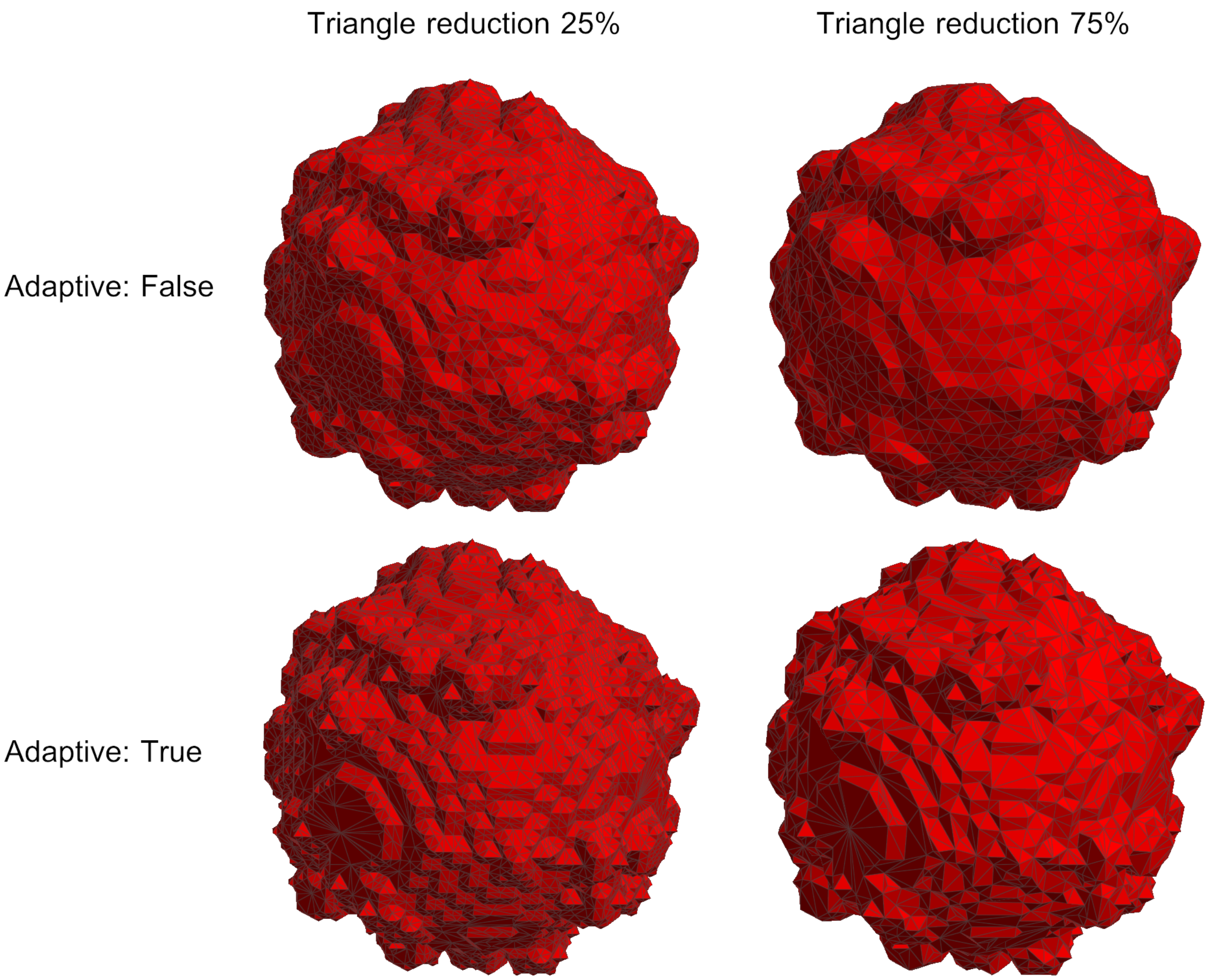
Target Number of Triangles

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

Adaptive

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

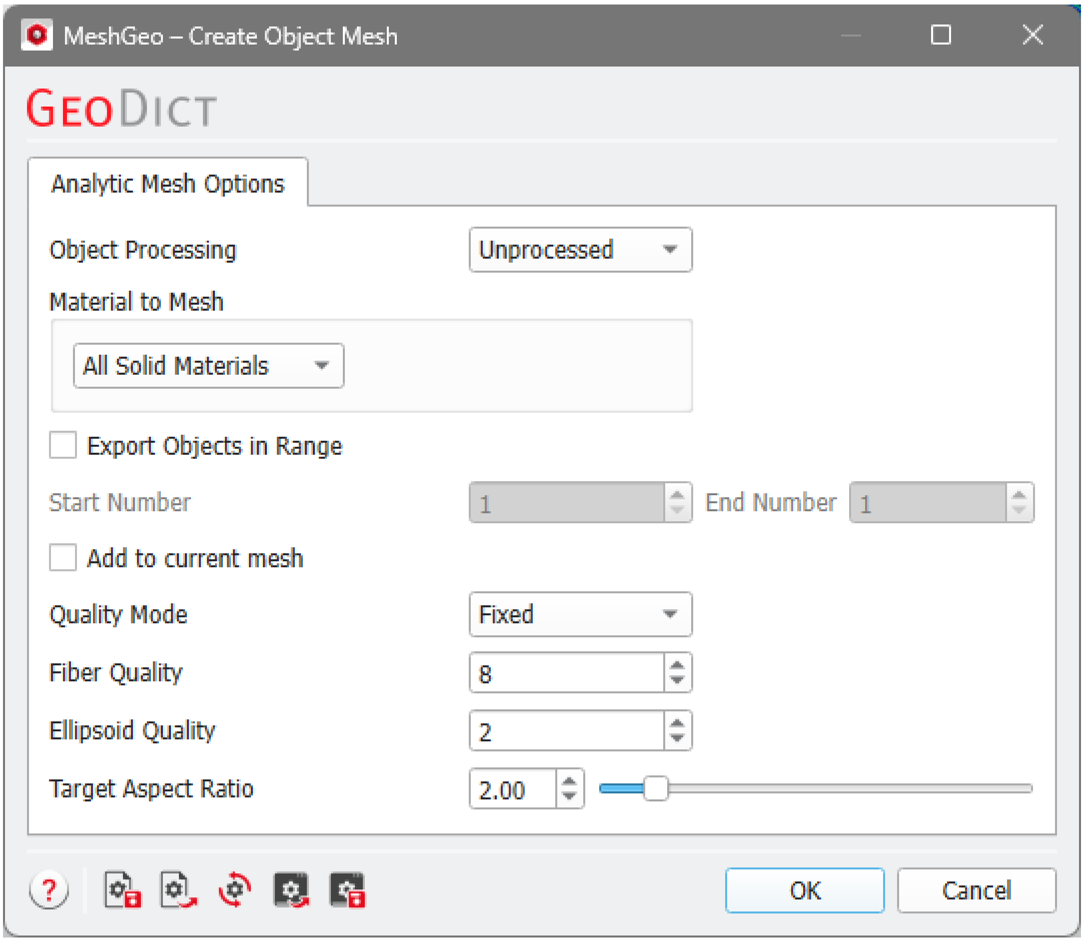
When the **Adaptive** mode is disabled, the coarsening algorithm works based on the edge lengths in the mesh.



1.1.2 Create Object Mesh

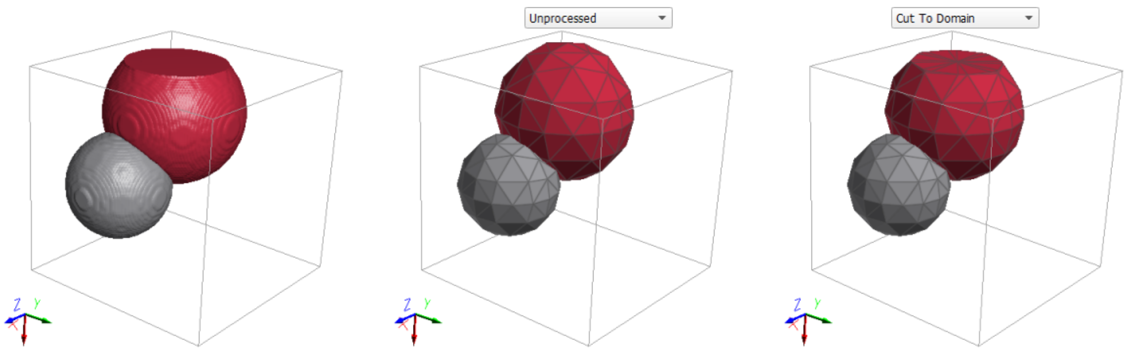
To export the analytic objects in the structure into a mesh, select **Create Object Mesh** from the pull-down menu. Each object is meshed separately, therefore, if analytic objects overlap, the meshes for these objects might overlap, too. If it is important that these meshes do not overlap, use the **Multi Material** 9 mode with **Export Voxel Mesh** instead.

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



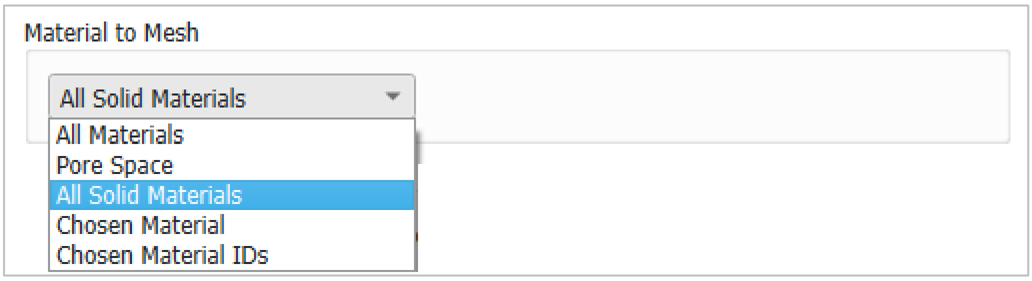
Object Processing

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

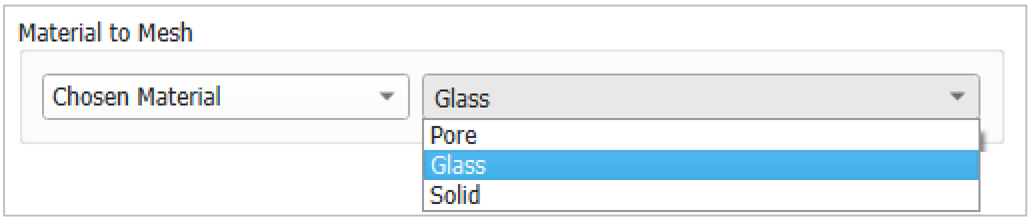


Material to Mesh

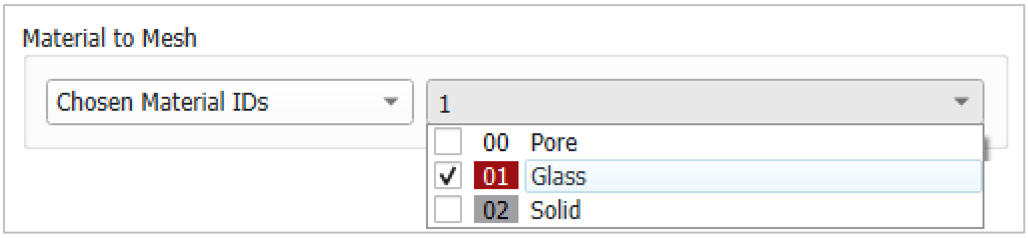
Select the Material IDs to be meshed with the options under **Material to Mesh**. With the first three options, either **All Materials**, the **Pore Space**, or **All Solid Materials** are meshed. **All Solid Materials** is the default, since this is the best choice for most use cases.



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



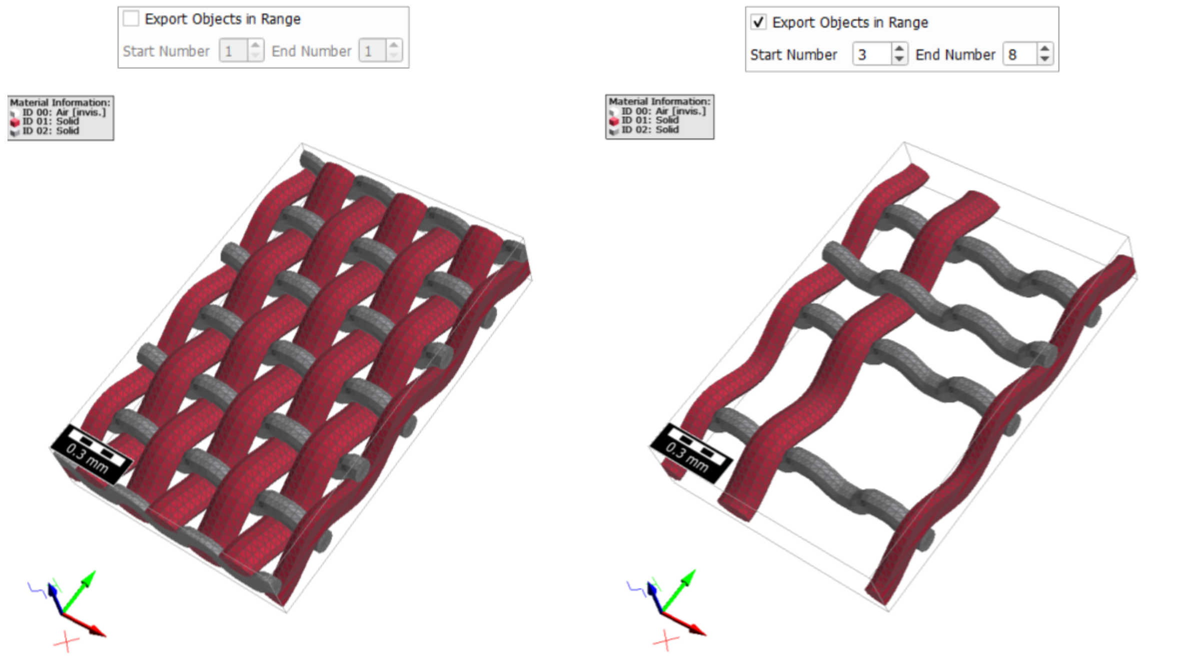
With **Chosen Material IDs**, individual Material IDs can be selected, which can have assigned different materials.



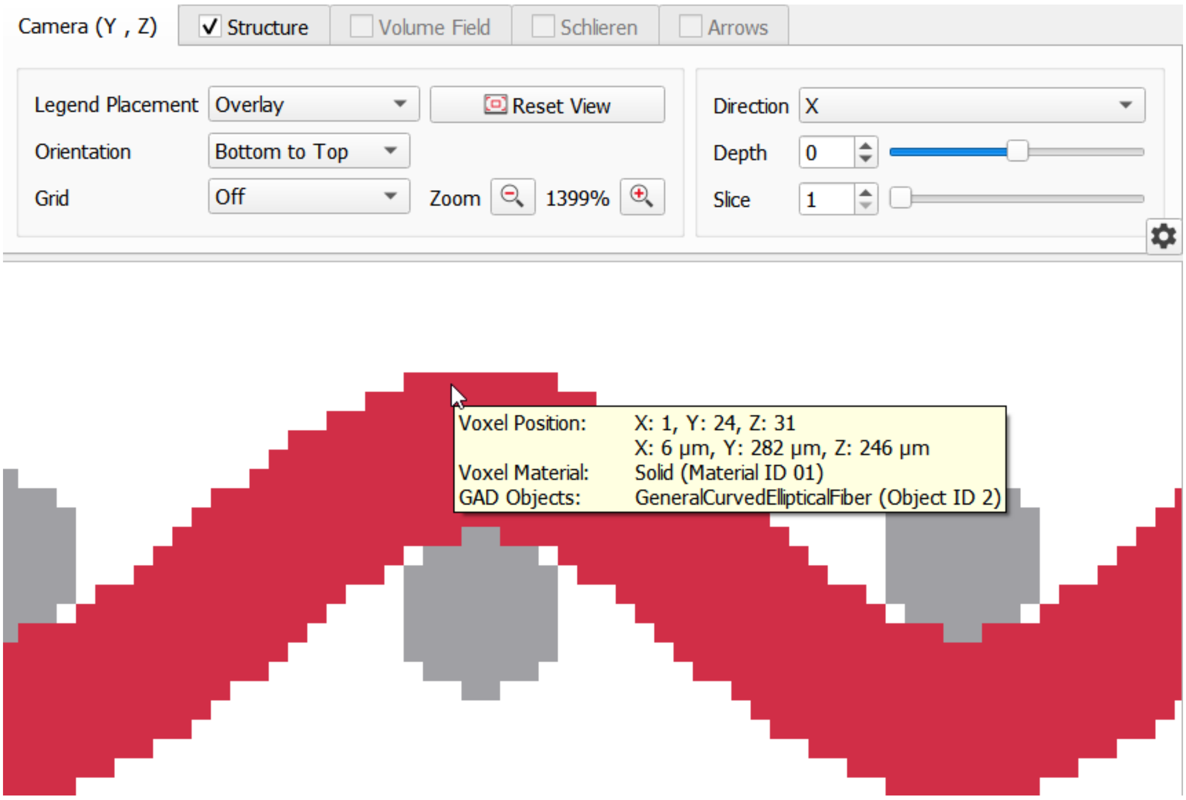
For **Analytic Mesh**, not all Material IDs in the structure can be used to create a mesh. For example, the pore space is not an object in GeoDict and therefore no analytic mesh can be created from it. Furthermore, the overlap between objects is not an analytic object itself and can therefore not be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

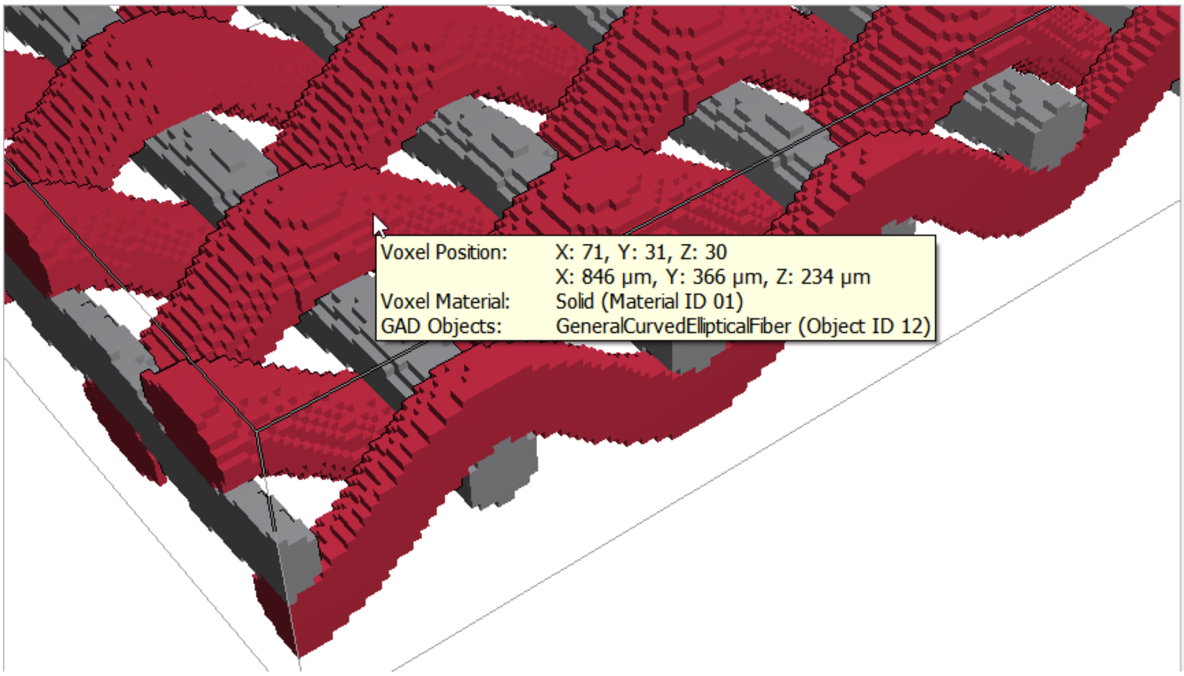
Export Objects in Range

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



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



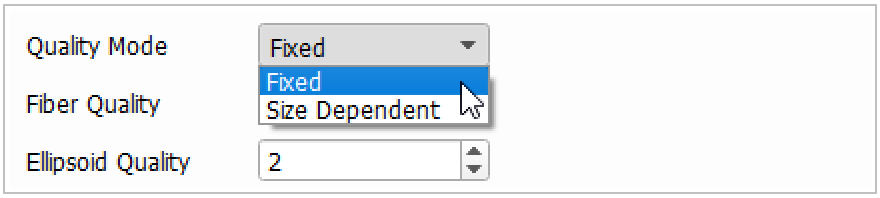


Add to current mesh

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 saved to a single file.

Quality Mode

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



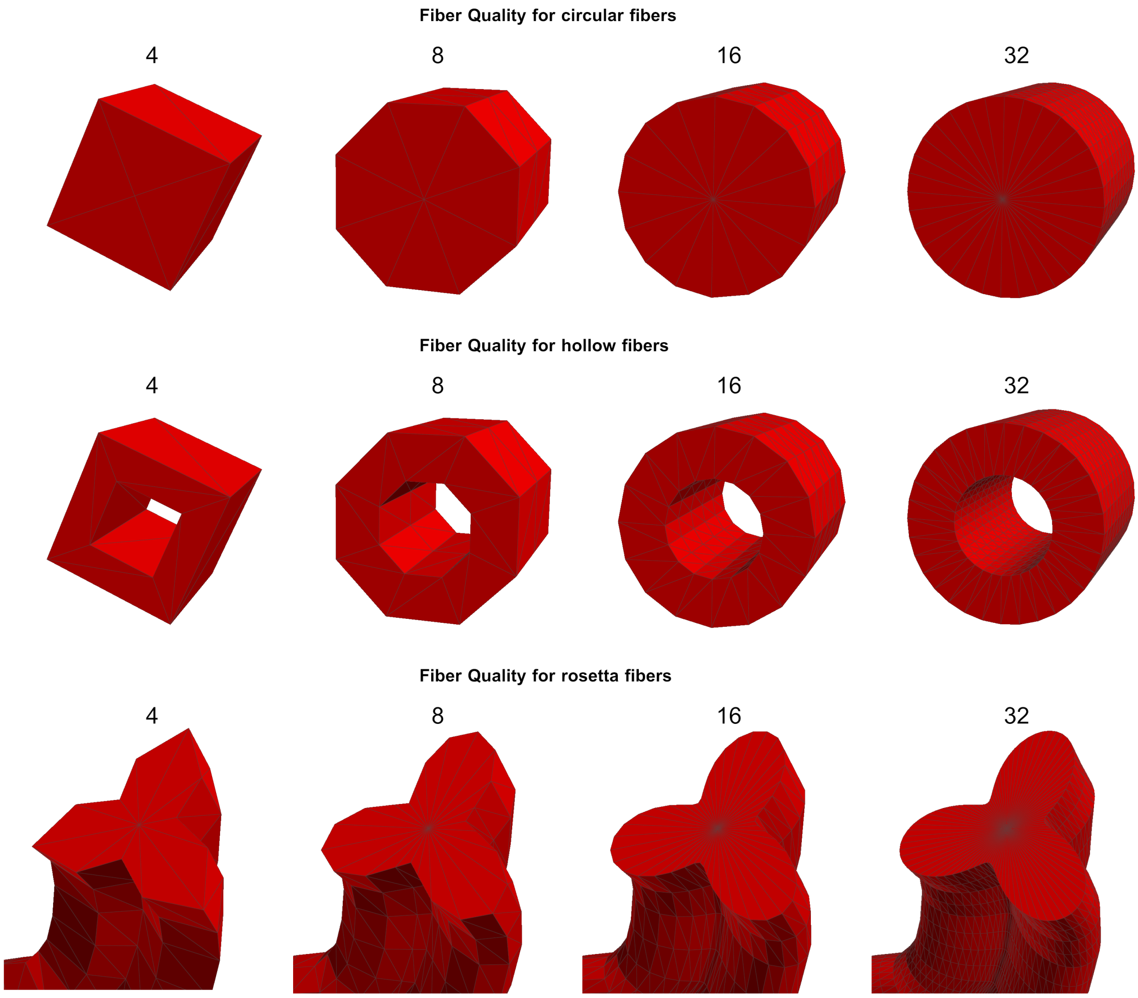
With **Quality Mode** set to **Fixed**, the **Fiber Quality** and **Ellipsoid Quality** are defined globally for the structure. When choosing **Size Dependent**, minimum and maximum values must be defined. This option is recommended if the size of the objects in the structure varies by a large amount.

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

Fiber Quality

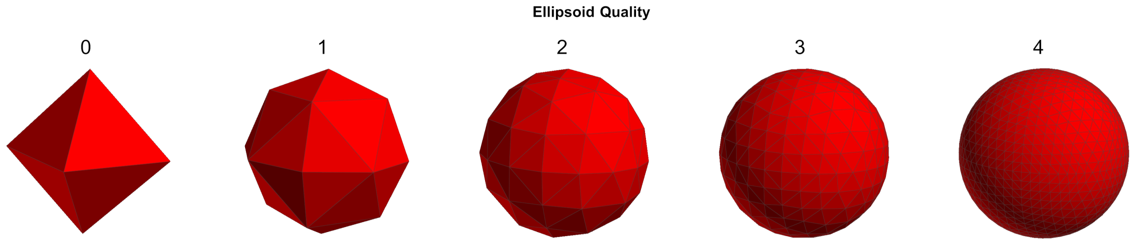
Fiber Quality controls the resolution for fibers. The entered value determines the number of nodes in which the profile shape is resolved.

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



Ellipsoid Quality

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



Target Aspect Ratio

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

1.2 Analyze

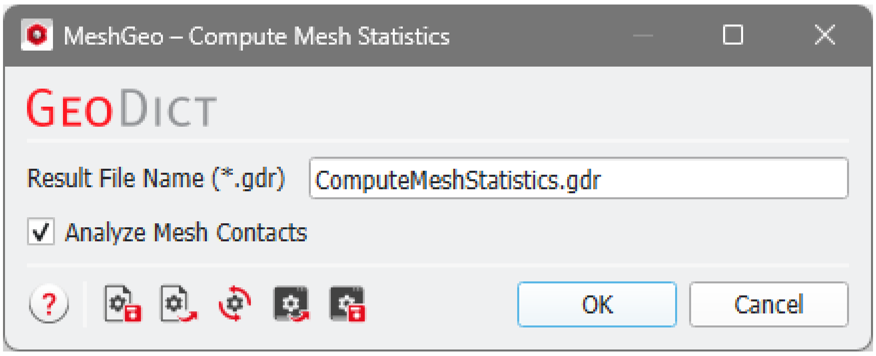
By visualizing the **Triangles**, the overall quality of the surface mesh can be determined. However, visual inspection does not capture fine details and makes it difficult to evaluate each individual triangle. The **Analyze** function offers a more detailed assessment of mesh quality through the options of **Compute Mesh Statistics** and **Compute Three-phase Contact Line**.

Commands of Analyze:

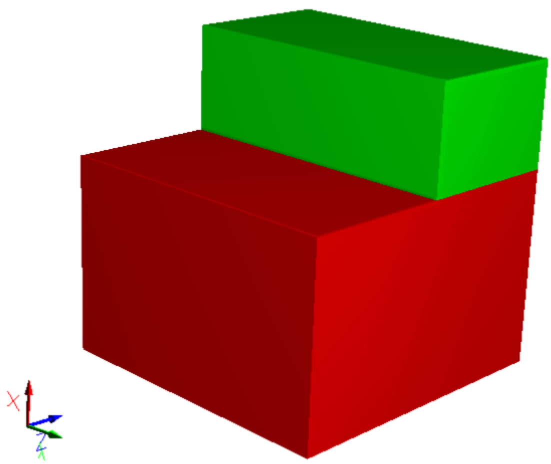
- Compute Mesh Statistics 22
  Compute the statistics of vertices, edges, faces, meshes, surface area, angle, coordinates, and more.
- Compute Three-Phase Contact Line 24
  Compute three-phase contact line at which three different materials meet.

1.2.1 Compute Mesh Statistics

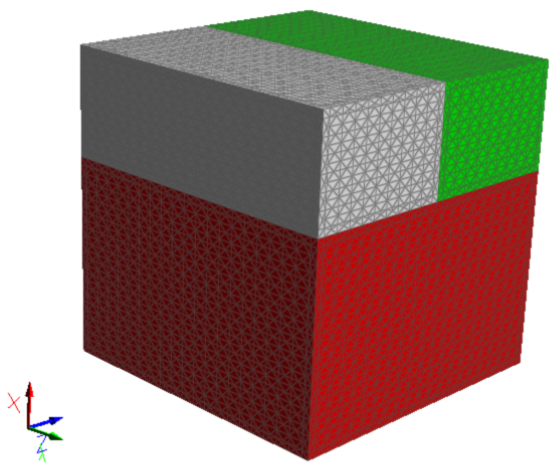
Click the **Edit...** button to set the parameters. The option **Analyze Mesh Contacts** determines if the shared interface between individual meshes should be analyzed or not.



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.

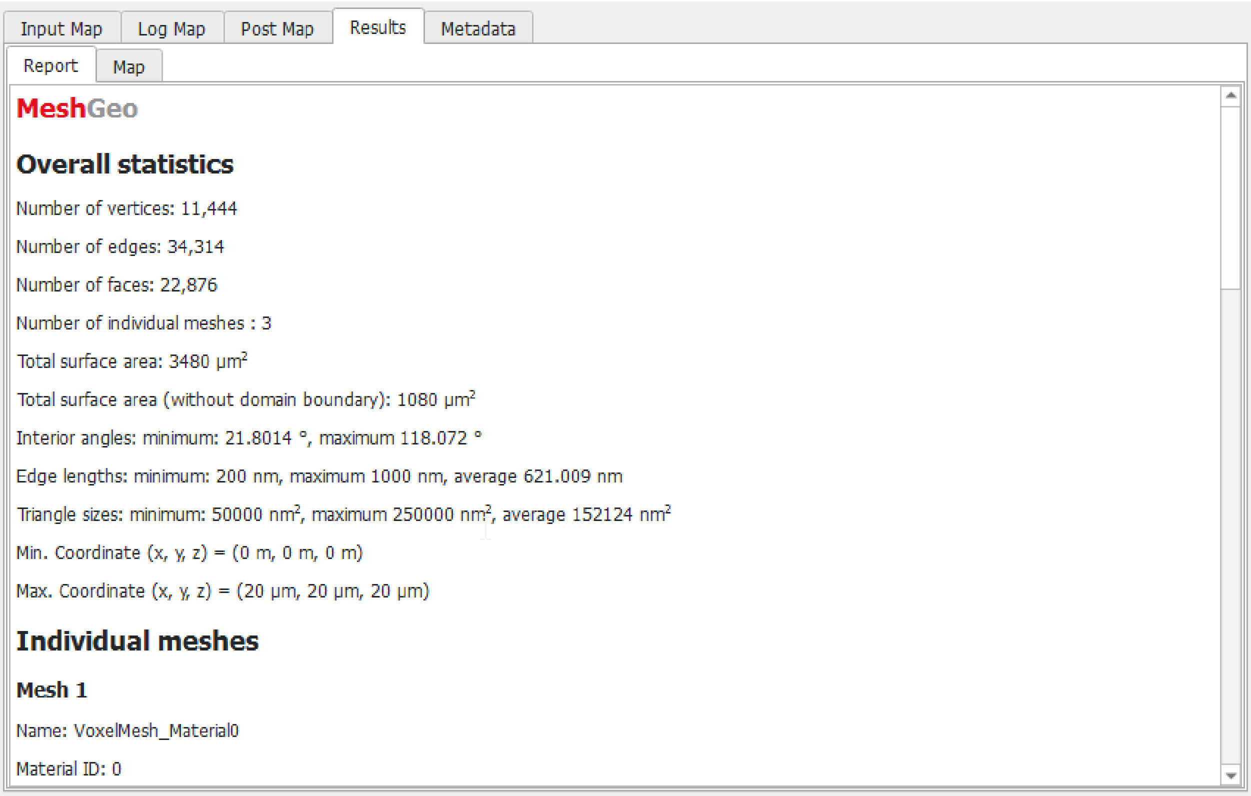


Voxel structure

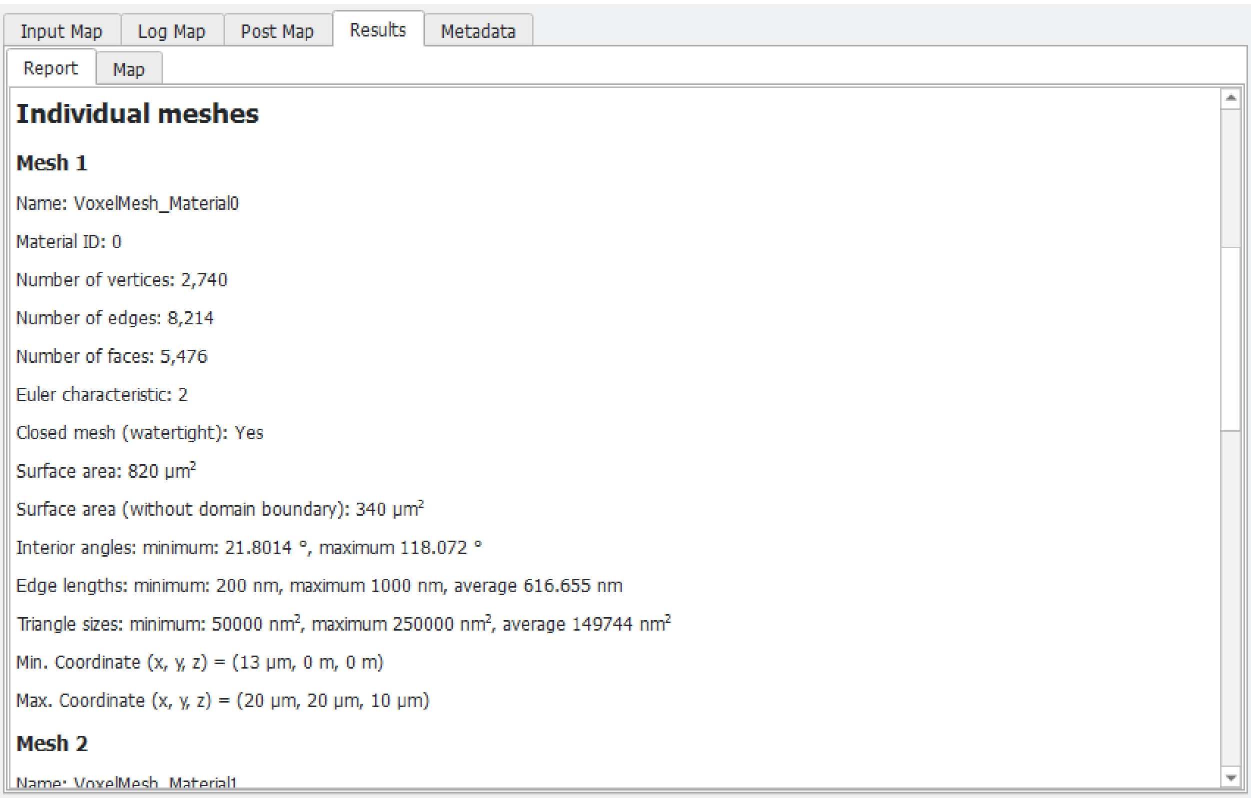


Meshes for all matertials

The overall statistics are shown in result viewer, include the number of vertices, edges, faces, meshes, surface area, angle, and coordinates.

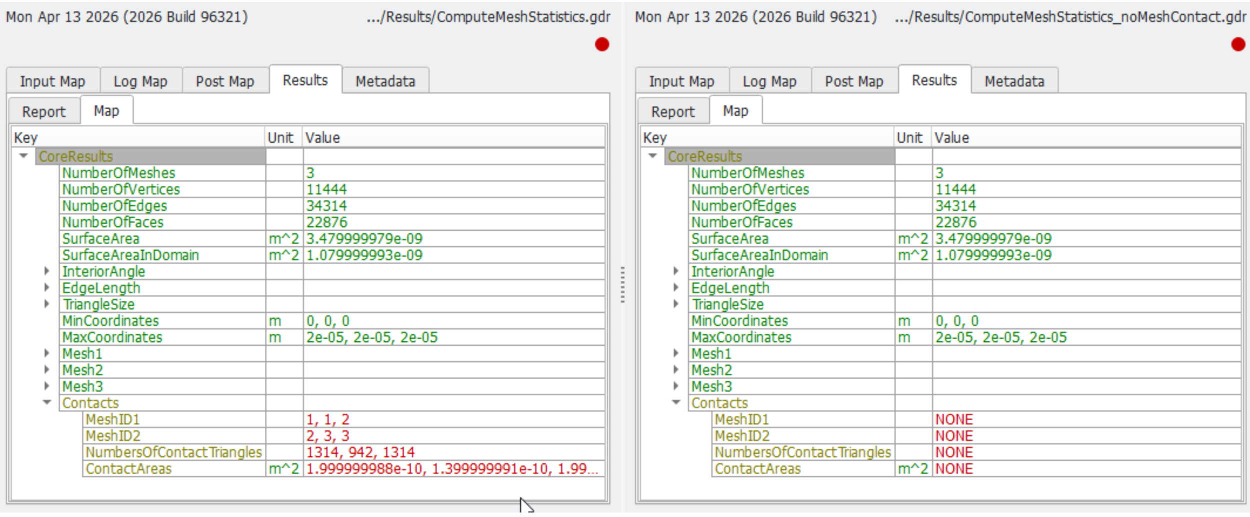


Detailed information is also given for each individual mesh.



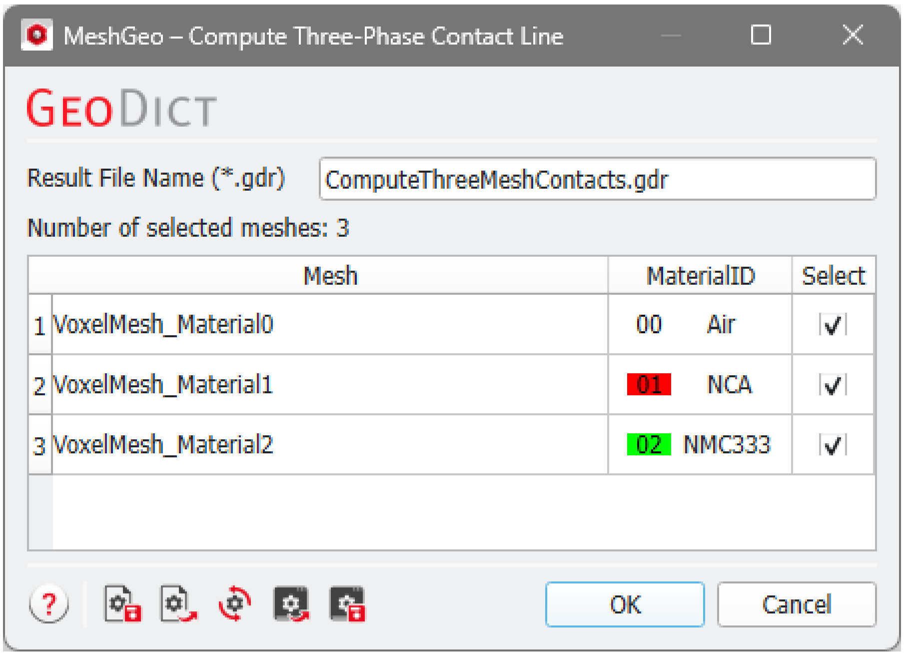
All the information can be found additionally in the result **Map** tab that can be accessed also with GeoPy scripts.

See the Result Viewer user guide for more information. When **Analyze Mesh Contacts** is not chosen, the information of **Contacts** is not provided.

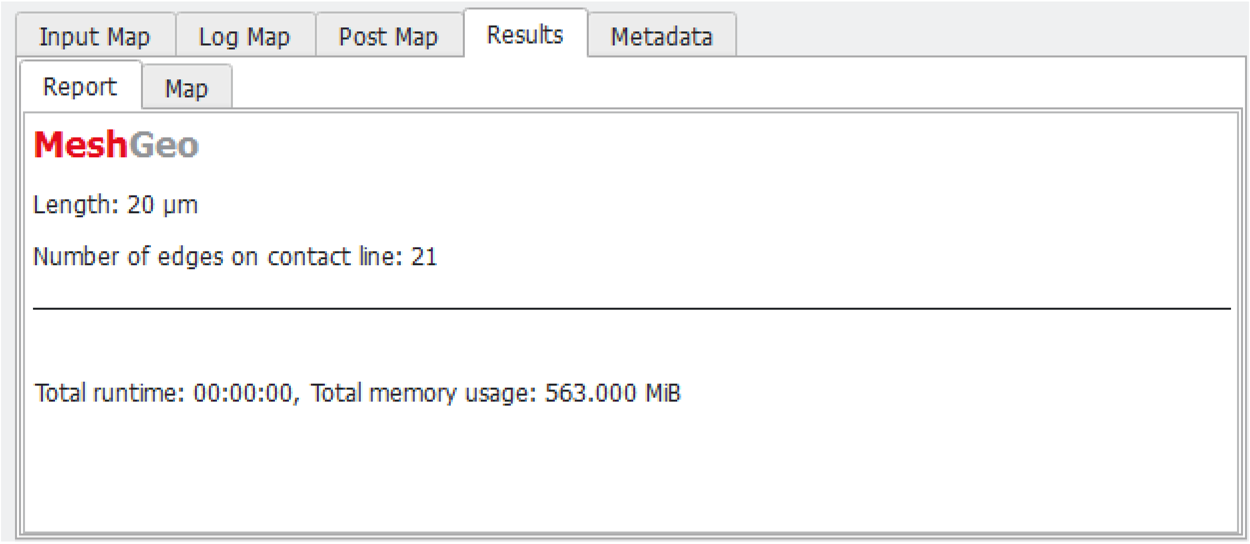


1.2.2 Compute Three-Phase Contact Line

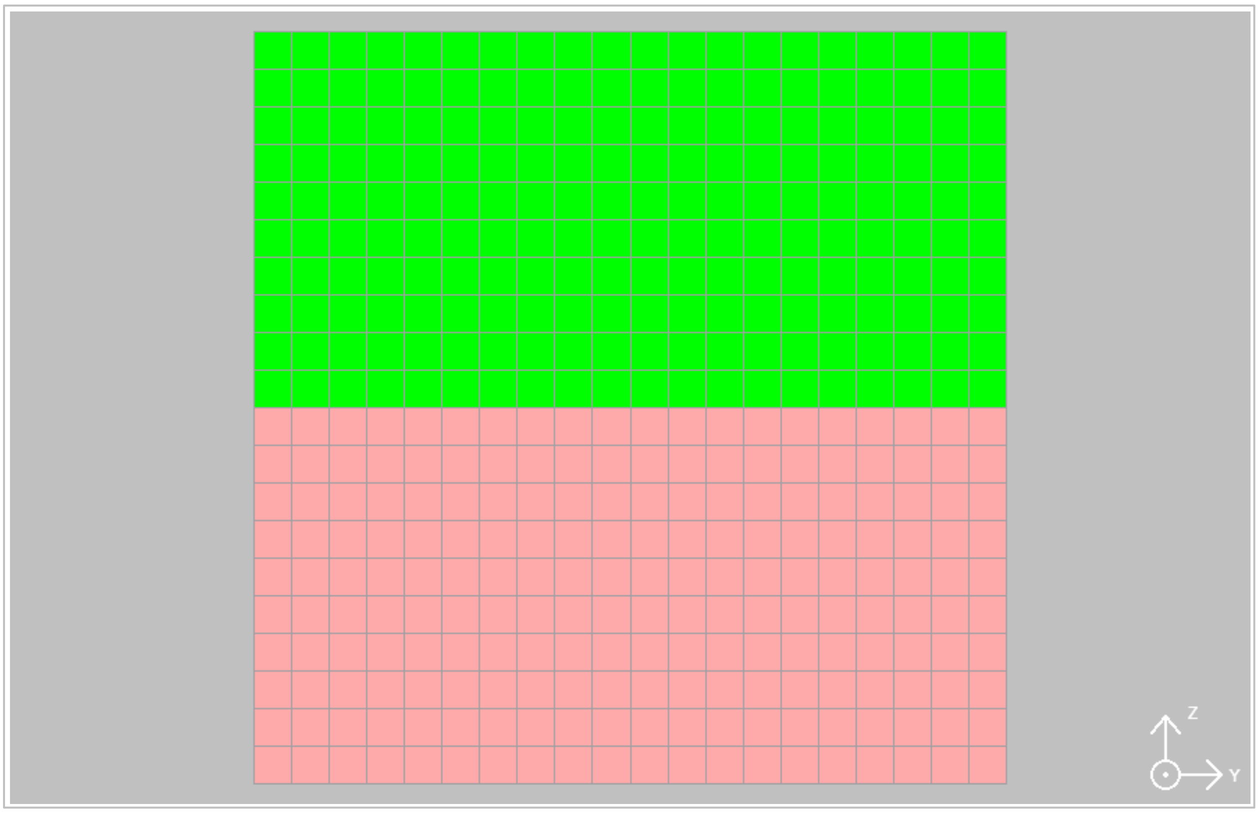
To describe the interface between different materials, the length the of three-phase contact line at which three different materials meet can provide helpful information. To compute a three-phase contact line, at least 3 meshes need to be loaded. For the brick example in the previous topic 22, we can choose the three meshes and start the computation.



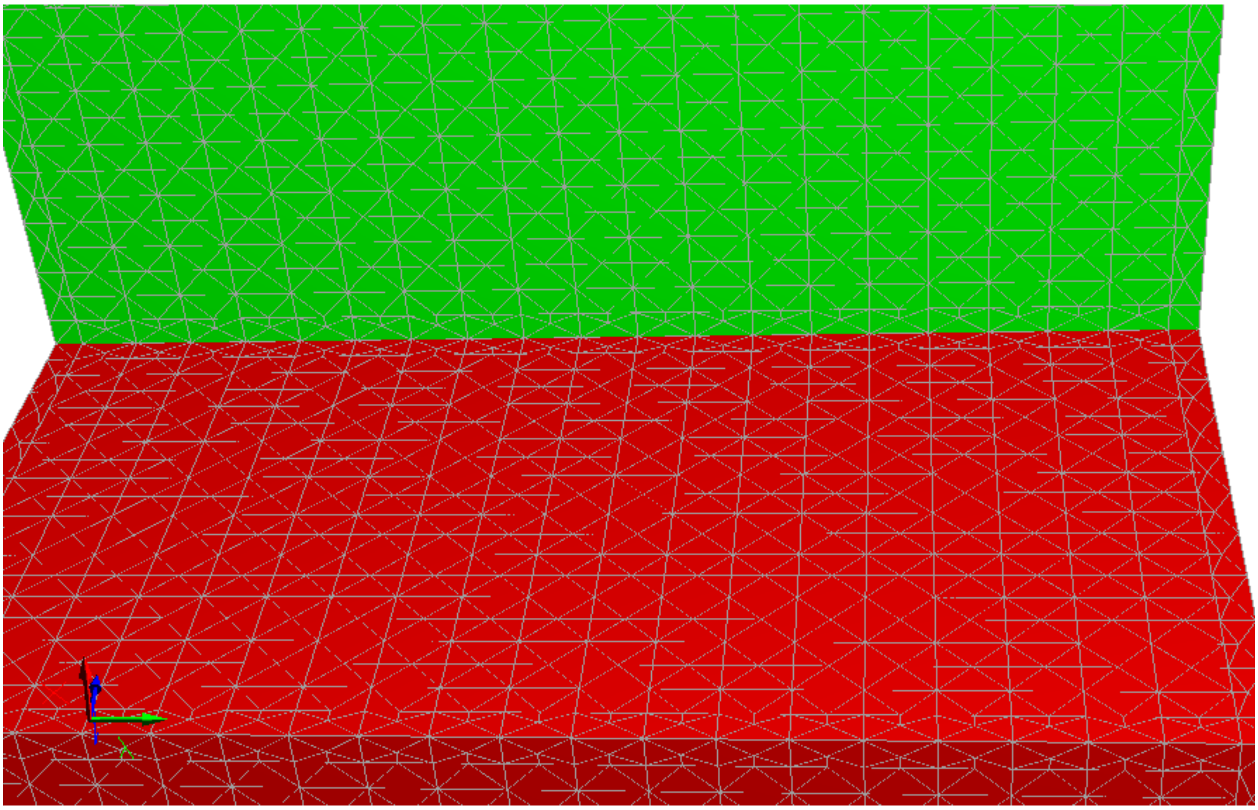
The length of the contact line and the number of edges is given.



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



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.



1.3 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**.

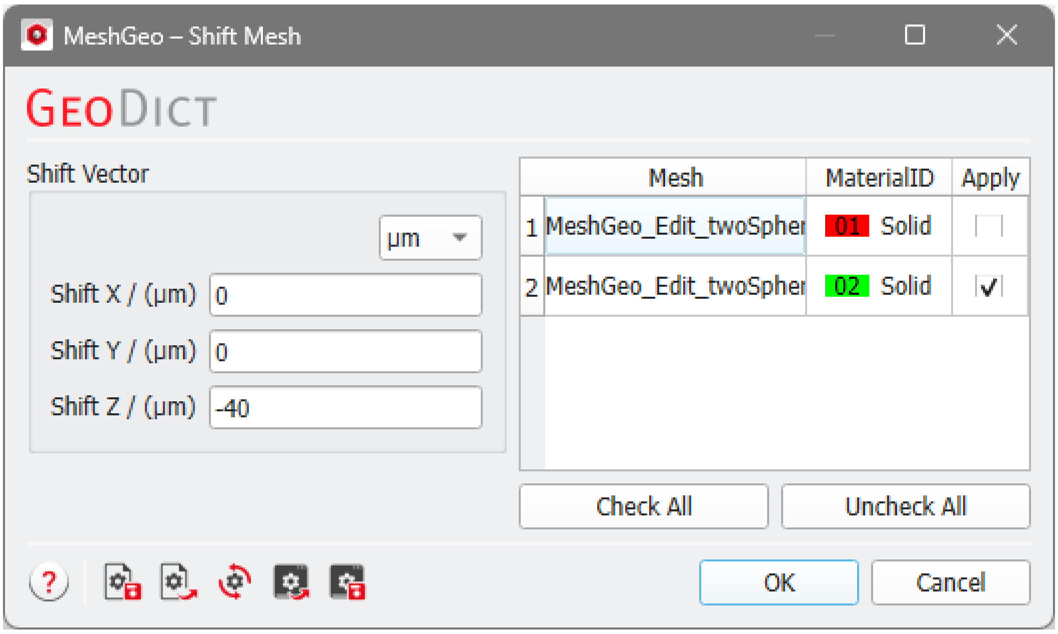
Commands of Edit:

- Shift Mesh 27
  Shift the specified meshes by a given distance.
- Scale Mesh 28
  Perform a vector-based scaling transformation on the specified meshes.
- Rotate Mesh 32
  Rotate the selected meshes by a specified angle around a chosen center.
- Reassign Mesh Material ID 33
  Change the Material ID for the selected meshes.
- Delete Selected Mesh 35
  Remove specified meshes.
- Delete All Meshes 35
  Remove all meshes in memory.
- Split Meshes into Connected Components 36
  Create separate meshes for each connected component of the loaded surface triangulation.

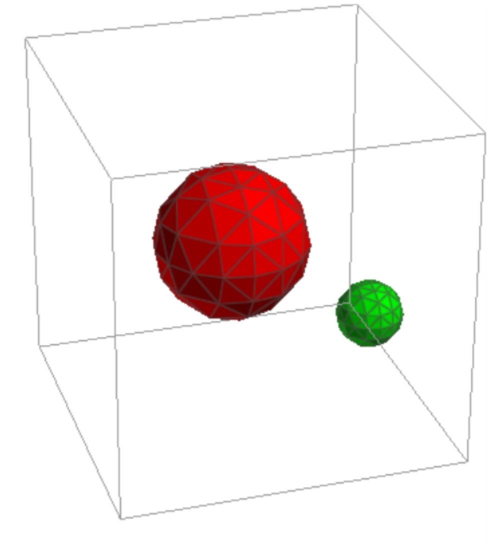
1.3.1 Shift Mesh

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

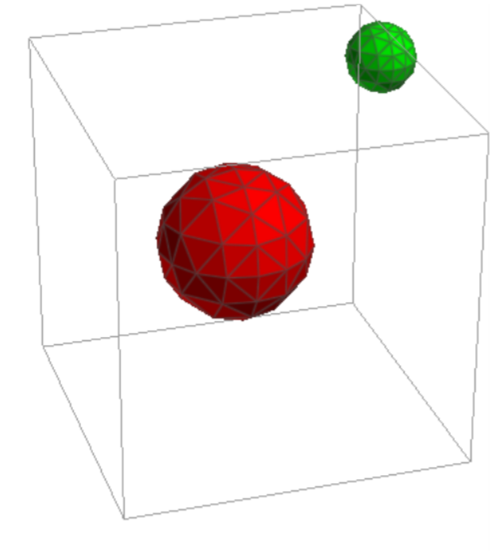
Select **Shift Mesh** and click the upper **Edit...** button (to the right of **Options**) to open the **Shift Mesh** dialog. Choose the mesh to be shifted in the table on the right and define the values for **Shift X**, **Shift Y**, and **Shift Z** for the shift distance in X, Y, and Z direction, respectively. 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 40 µm.



before shift



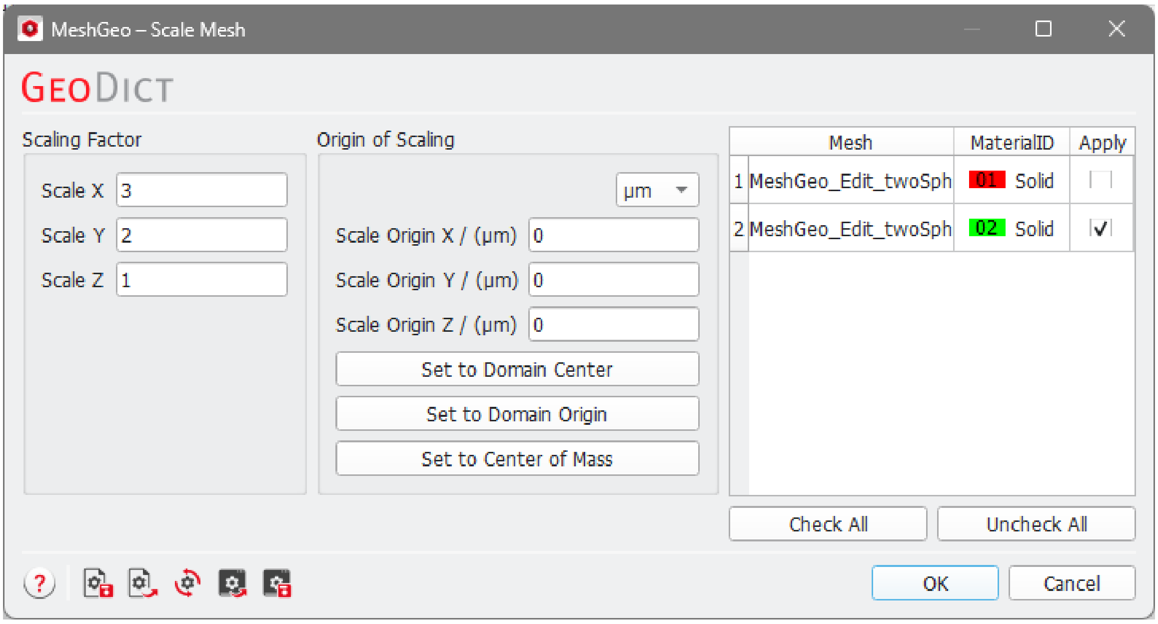
after shift

1.3.2 Scale Mesh

The mesh can be scaled with **Scale Mesh**. Select **Scale Mesh** and click **Edit...** to open the dialog of **Scale Mesh**.

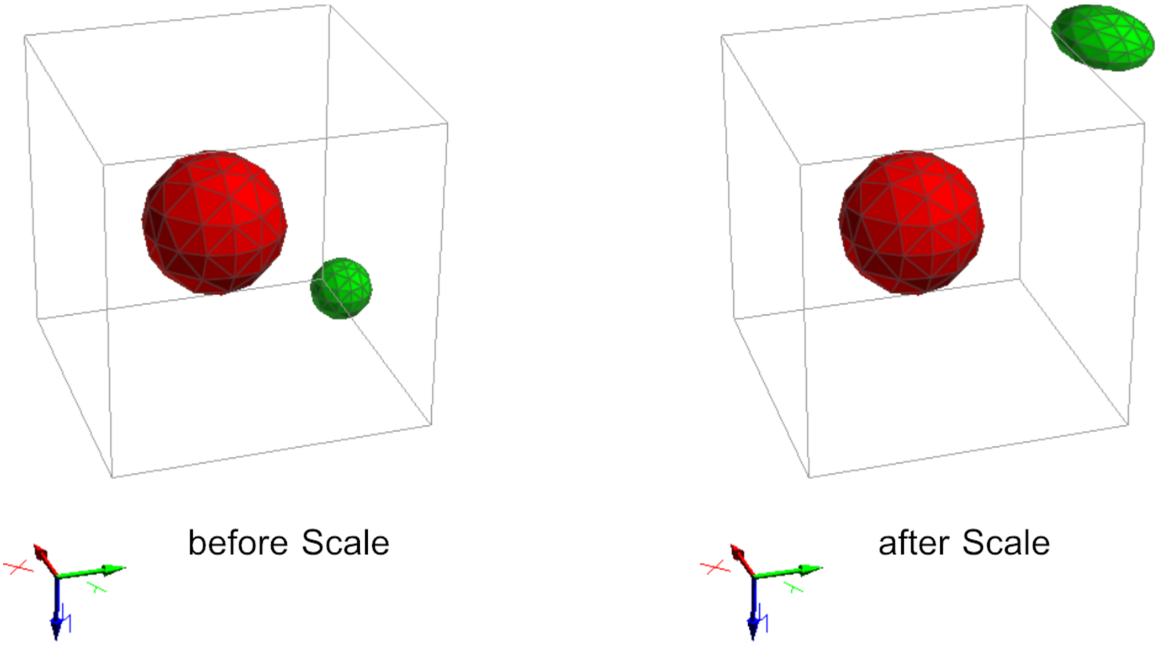
Choose the meshes to be scaled on the right box and define the **Scaling Factor** and the **Origin of Scaling**.

Scaling Factor



The scaling factors for the X, Y, and Z directions in **Scale X**, **Scale Y**, and **Scale Z** are separately given.

For the two-sphere example, the scaling factors of 3, 2, 1 for X, Y, and Z are given for the green ball with the domain origin, which is (0, 0, 0), as the scaling origin.



After scaling, not only the size of the object is changed, but also its center is shifted by the same factor (See the green particle before and after scaling for reference) when the domain origin instead of the center of the object is used.

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.

The screenshot below shows the coordinates of an example STL before 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 +5.464098e-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
```

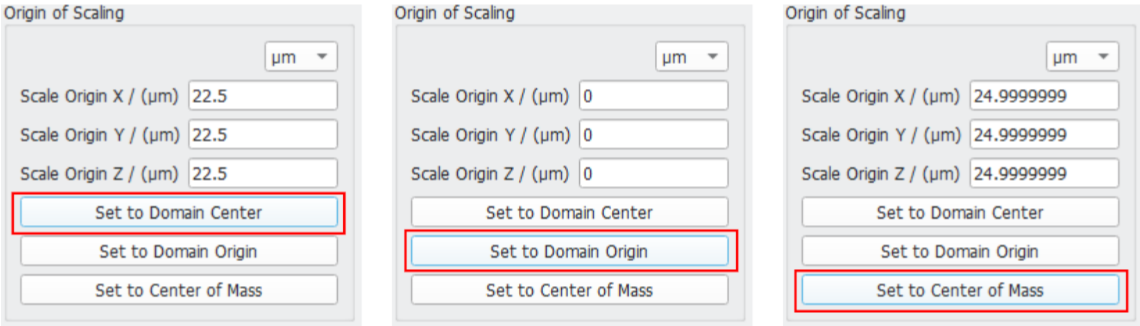
After scaling, the values of X-coordinate become 3 times of the original values, Y-coordinates become 2 times, and Z-coordinates keep the same as before.

```
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
```

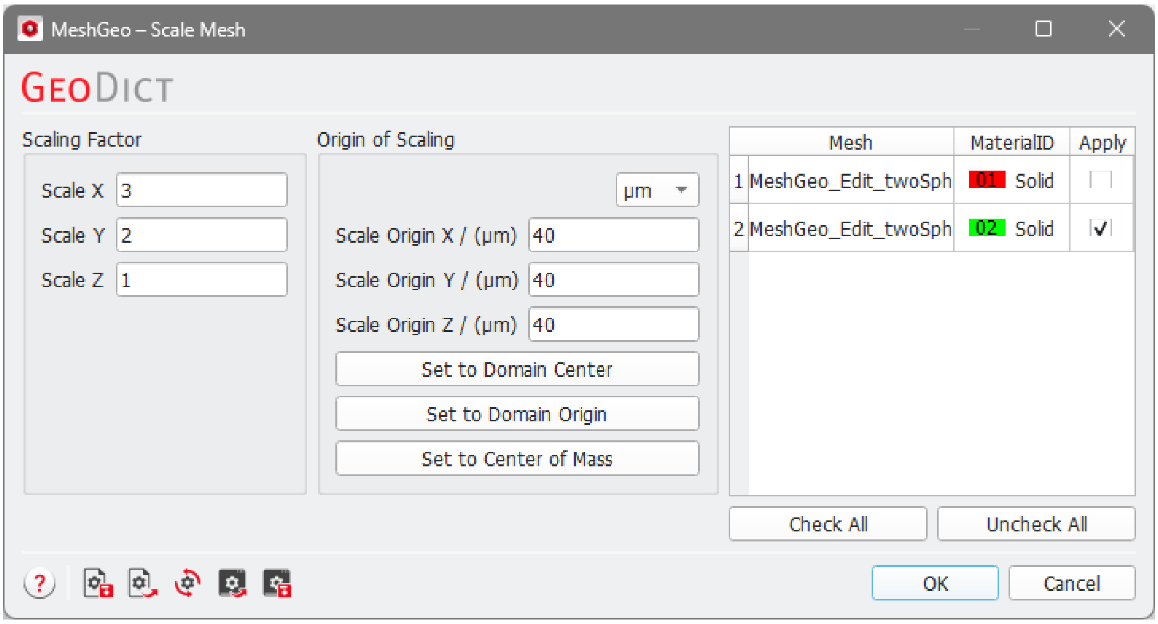
Origin of Scaling

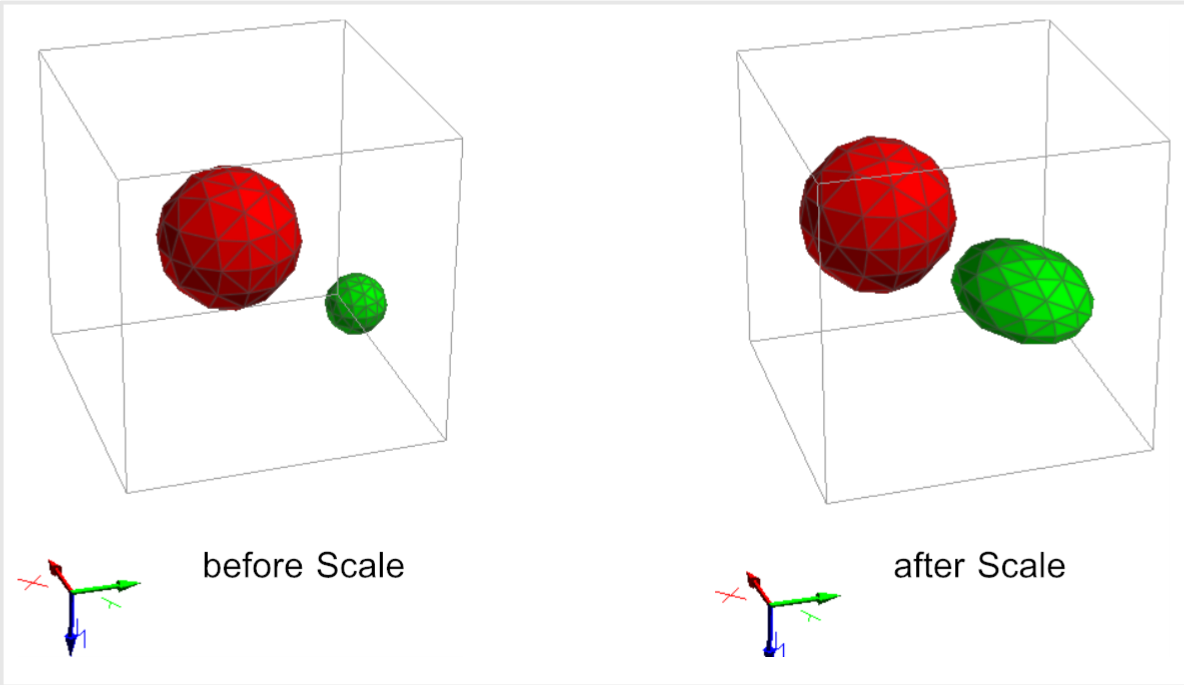
Prior to GeoDict 2026, the scaling origin was always anchored to the domain origin (0, 0, 0). Starting with GeoDict 2026, however, users can now customize and choose the **Origin of Scaling**.

For the example used above, the domain size is 200 × 200 × 200 voxels with the voxel length 0.225 µm, which means the physical domain has the size of 45 µm × 45 µm × 45 µm. When clicking **Set to Domain Center**, **Set to Domain Origin**, and **Set to Center of Mass**, the X, Y, and Z position of the scale origin is automatically shown. For the center of mass, the density and volume of each material are considered.



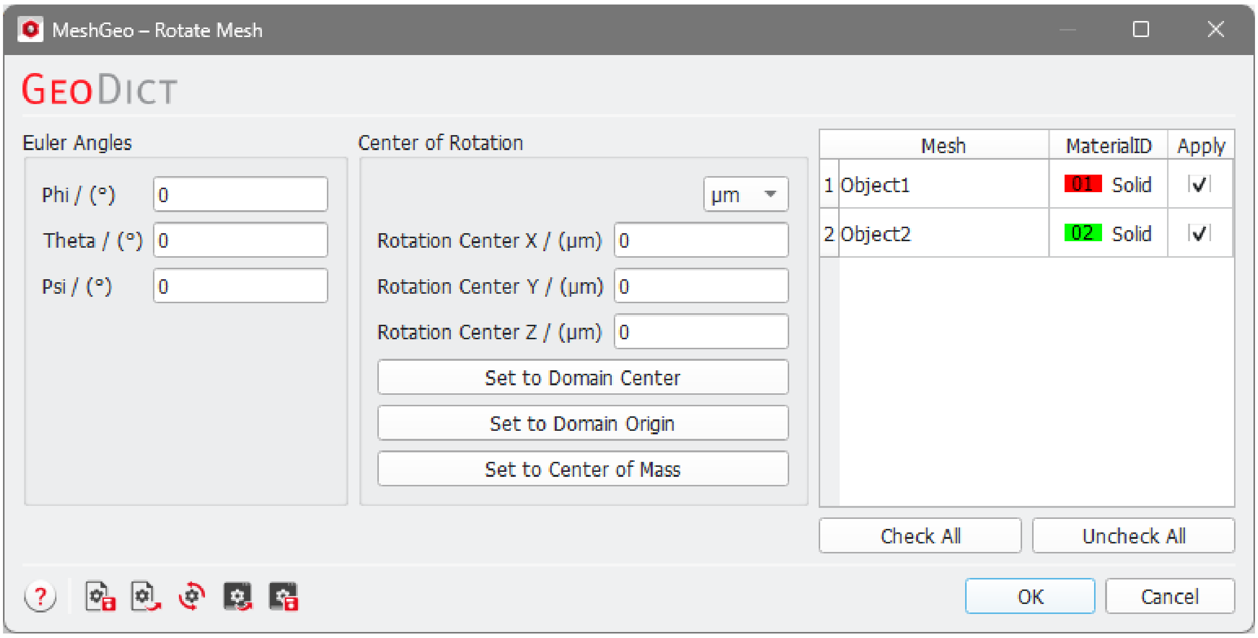
The **Scale Origin X**, **Scale Origin Y**, and **Scale Origin Z** can be manually given, too. For this example, when the center of the green ball is chosen for the **Origin of Scaling**, which should be manually determined, the ball center does not move, but only the shape changes.





1.3.3 Rotate Mesh

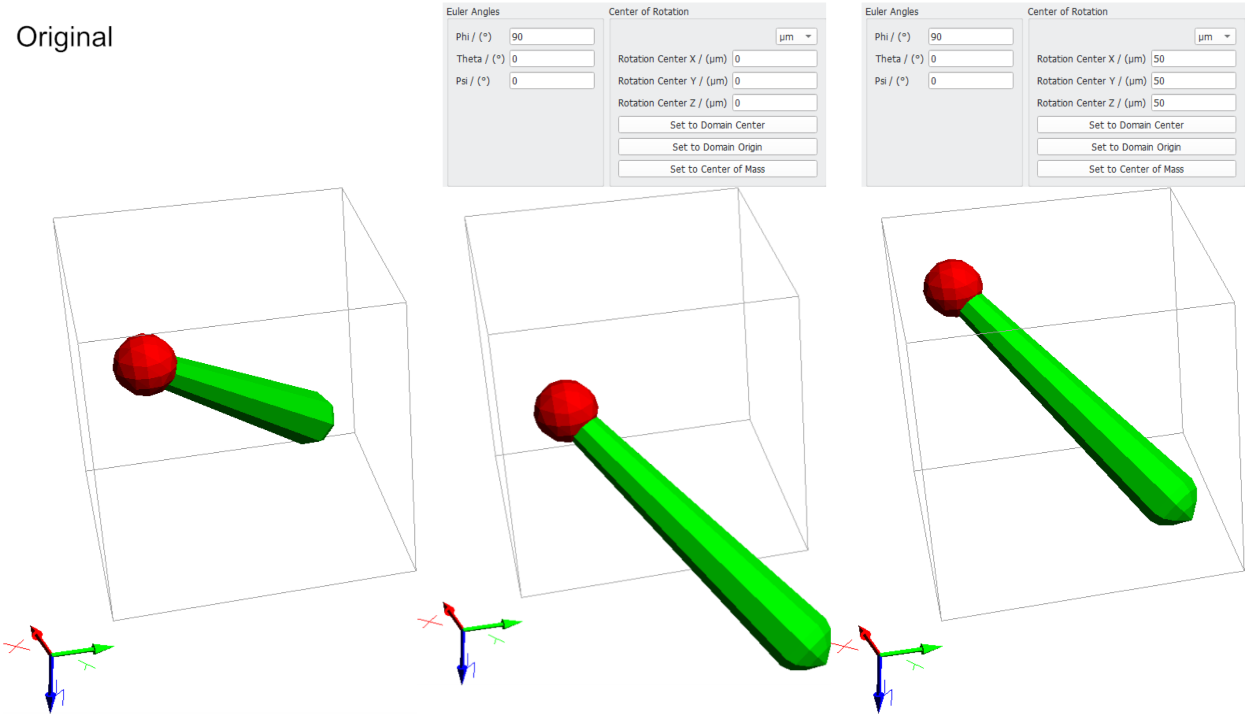
If a mesh needs to be rotated, select **Rotate Mesh** from the pull-down menu and click **Edit...** to open the **Rotate Mesh** dialog.



Enter the Euler Angles **Phi**, **Theta**, **Psi** by which the selected mesh should rotate. More details about Euler Angles can be found in the GadGeo chapter. The rotation can be around the chosen **Center of Rotation**. See the explanations about **Set to Domain Center, Set to Domain Origin, and Set to Center of Mass** 30.

An example of mesh rotation is shown below. The image on the left shows the original mesh of the match structure. The image in the middle shows mesh after the rotation of Phi with 90°

around the **Domain Origin**, and the right image shows the rotation of Phi with 90° around the **Domain Center**.

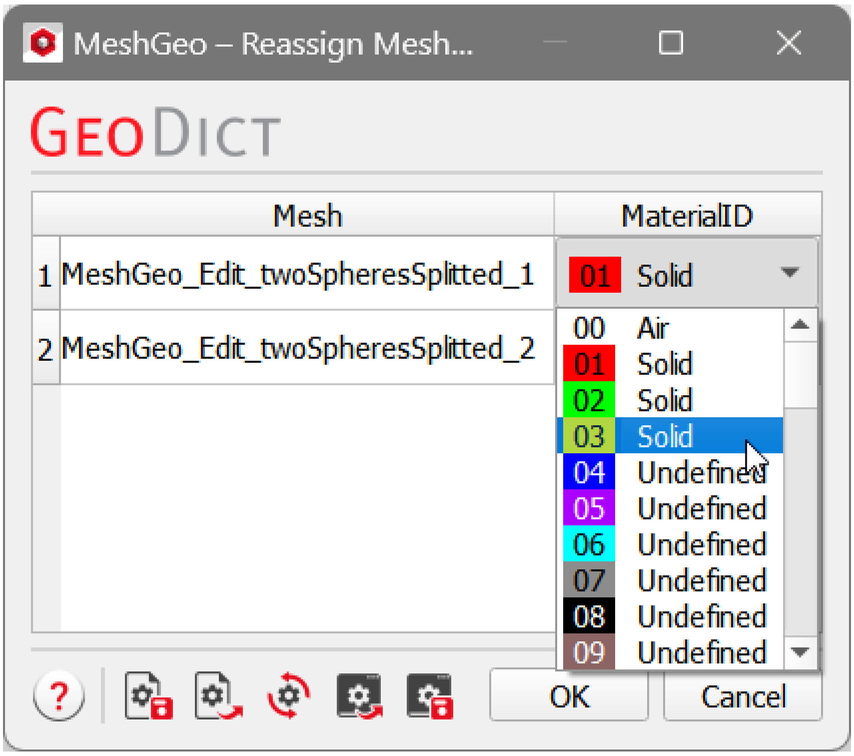


1.3.4 Reassign Mesh Material ID

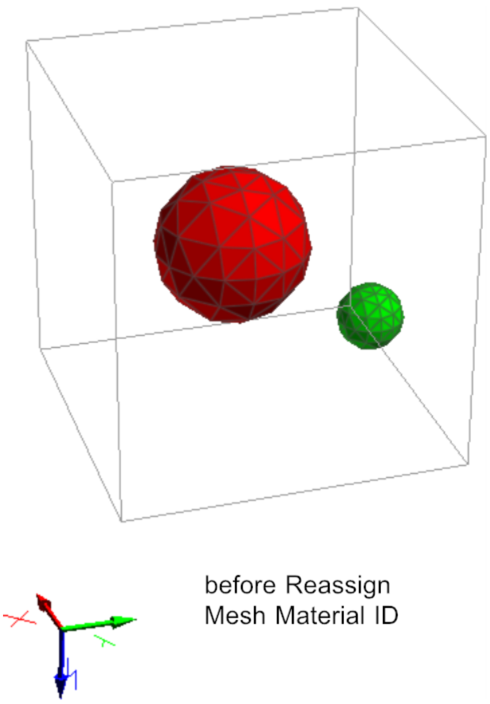
Each mesh can be assigned to a different Material ID with **Reassign Mesh Material ID**.

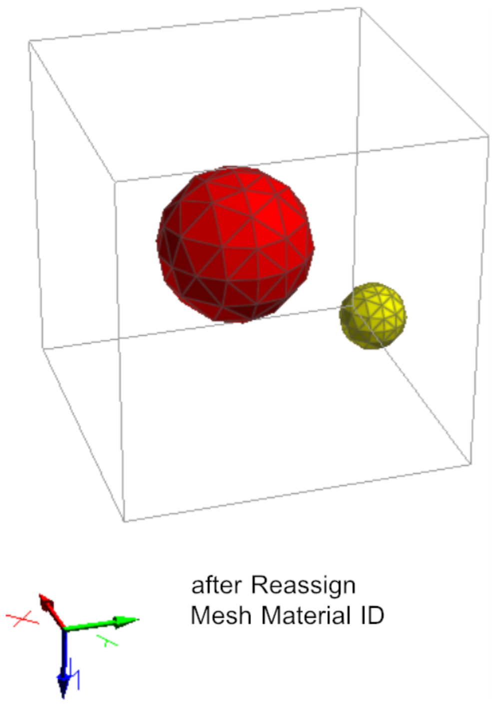
> **Reassign Mesh Material ID** is often used after Split Meshes into Connected Components 36, where meshes are splitted for different components but a same Material ID is used.

Click **Edit...** to open the dialog.



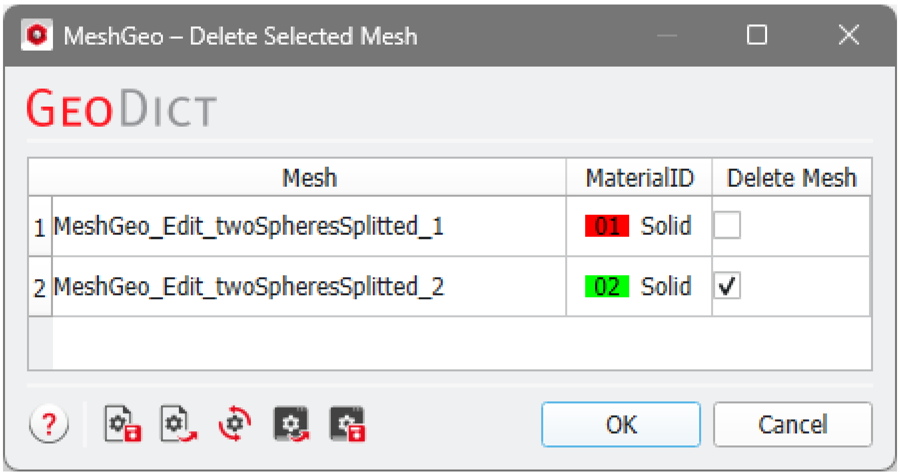
In the **Mesh** column, the currently loaded meshes are listed and the corresponding Material IDs are shown on the right. Each of the Material IDs can be reassigned. For the example below, the Material ID of the smaller sphere is originally 02, we change it to Material 03 and click **OK**. The Material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





1.3.5 Delete Selected 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.

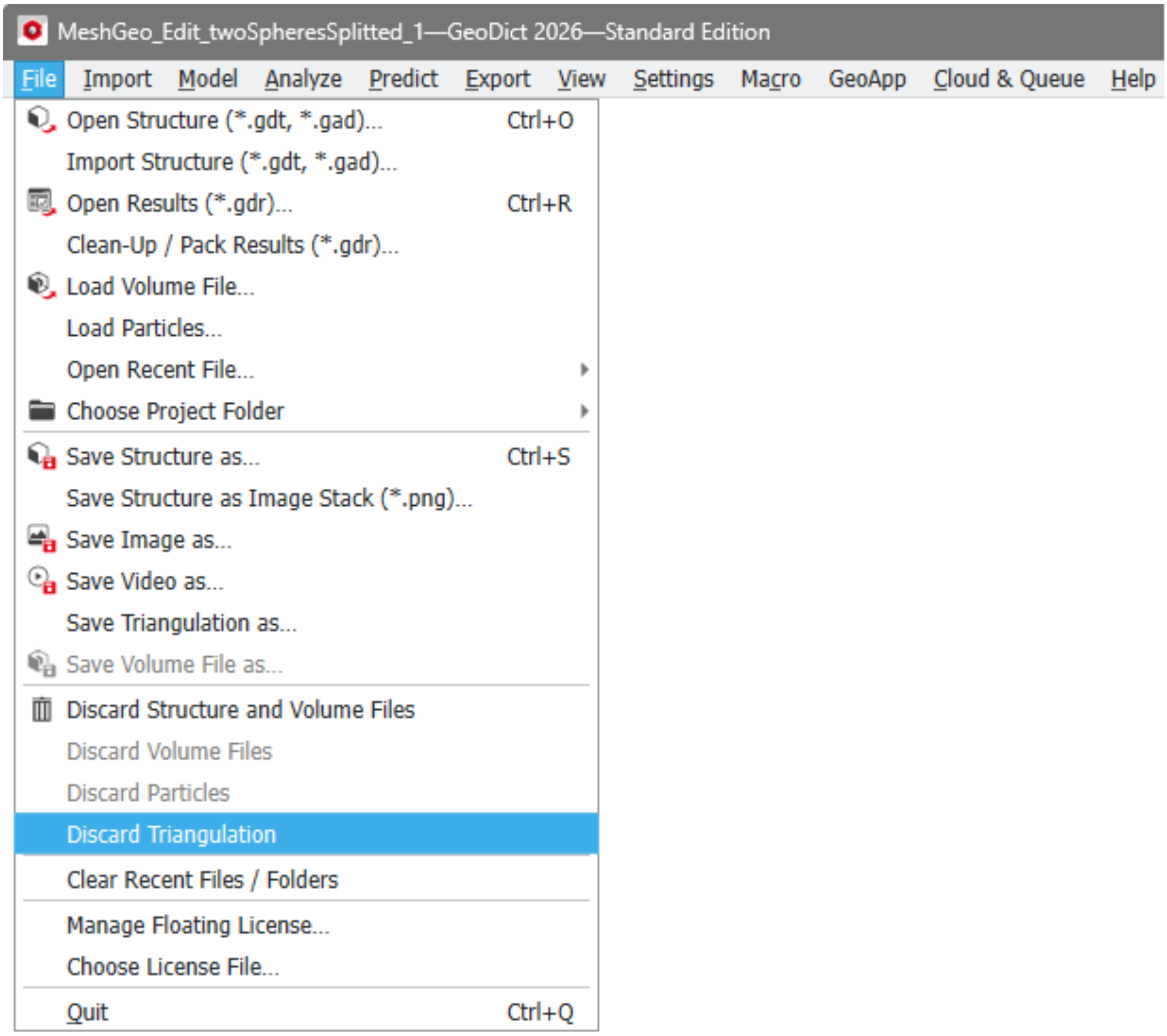


Choose the meshes to be deleted by clicking the checkboxes in the **Delete Mesh** column, then click **OK** to go back to the MeshGeo section. Click **Edit** to remove the selected meshes.

1.3.6 Delete All Meshes

Delete All Meshes deletes all the surface triangulations currently in memory. No options need to be set. Click **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.

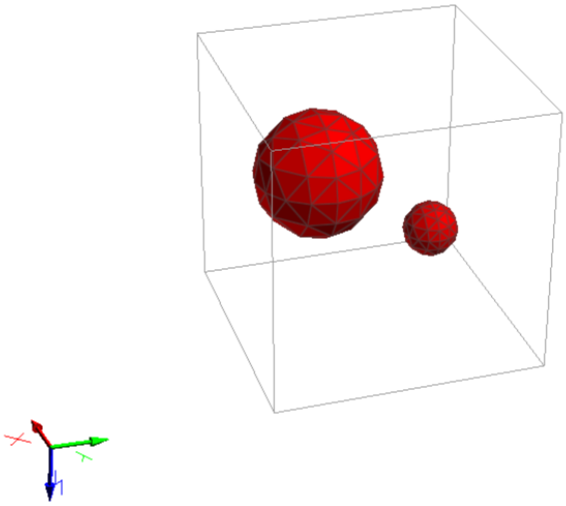


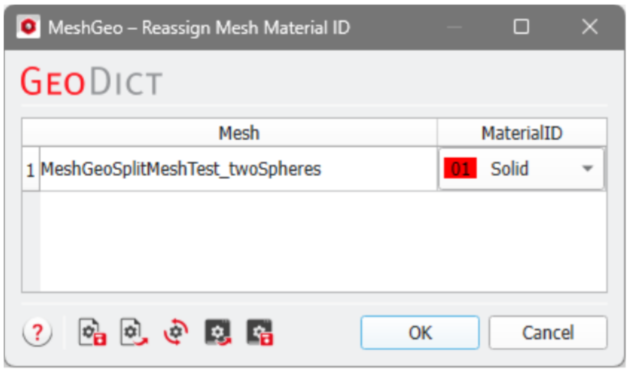
1.3.7 Split Meshes into Connected Components

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, with a single mesh handle for all the objects.

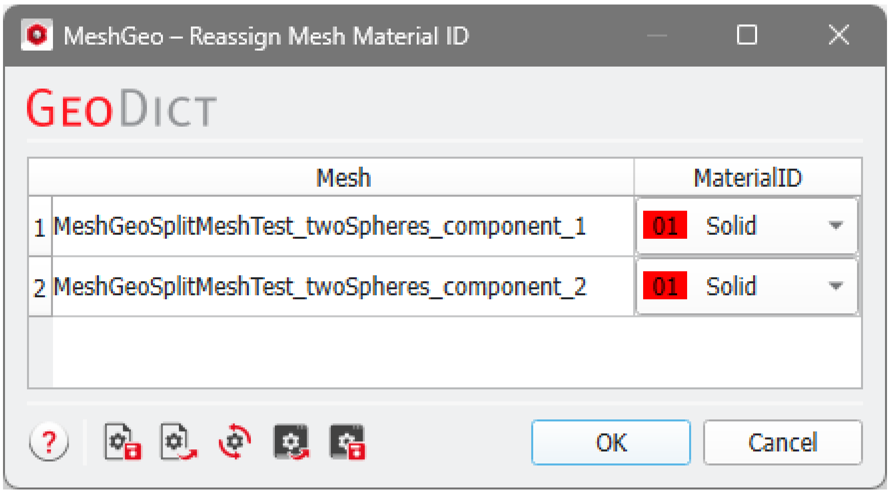
Open the settings dialog for **Reassign Mesh Material ID** 33 and observe that only one mesh is shown.





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**).

1.4 Smooth, Coarsen & Remesh

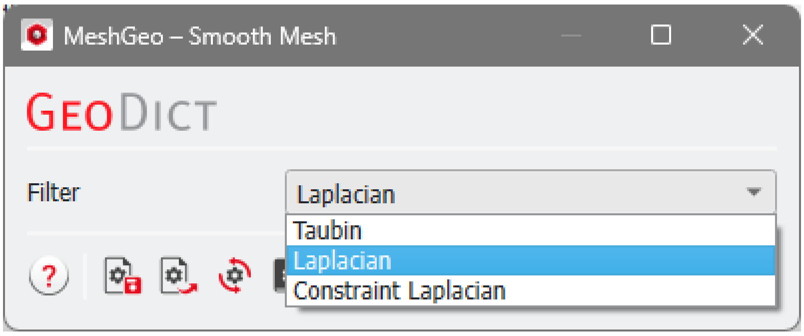
The currently loaded triangulation can be smoothened, coarsened, as well as remeshed by choosing the corresponding options under **Smooth, Coarsen & Remesh**.

Commands of Sooth, Coarsen & Remesh:

- Smooth Mesh 38
  Reduce surface roughness.
- Coarsen Mesh 39
  Reduce the number of triangles.
- Remesh Mesh 42
  Generate a new mesh.

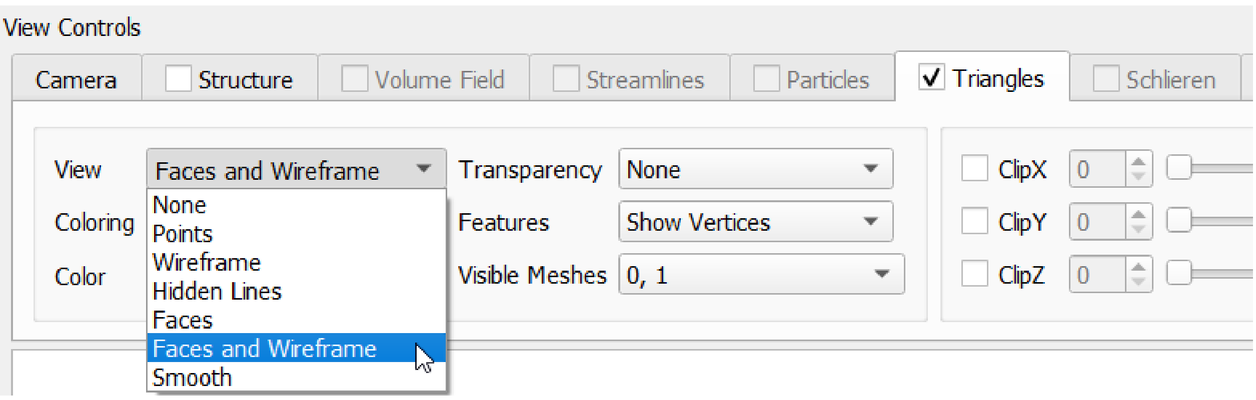
1.4.1 Smooth Mesh

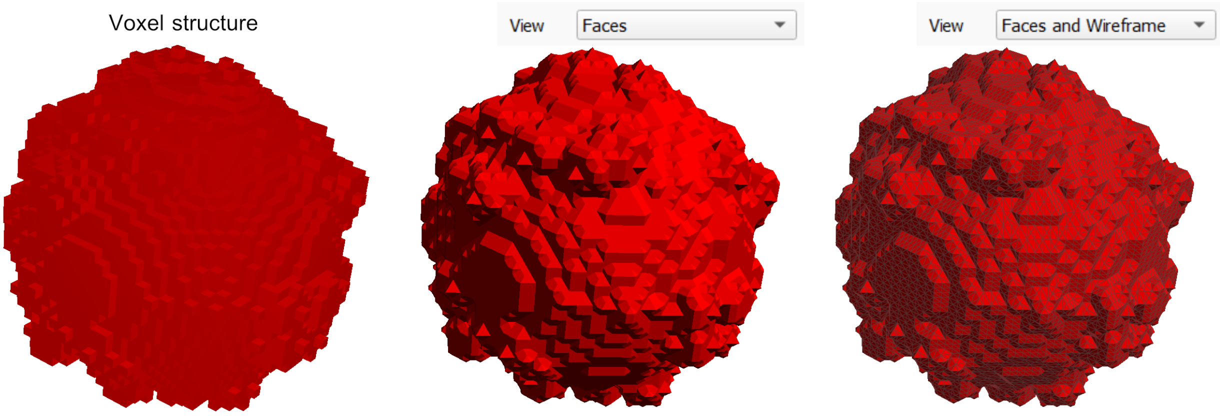
Meshes from voxel structures often have a significant surface roughness which can be reduced by applying smoothing filters. In MeshGeo, three smoothing filters are selectable from the pull-down menu to improve the mesh quality: **Taubin**, **Laplacian**, or **Constraint Laplacian**. Choose a filter and 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 9 mode.

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



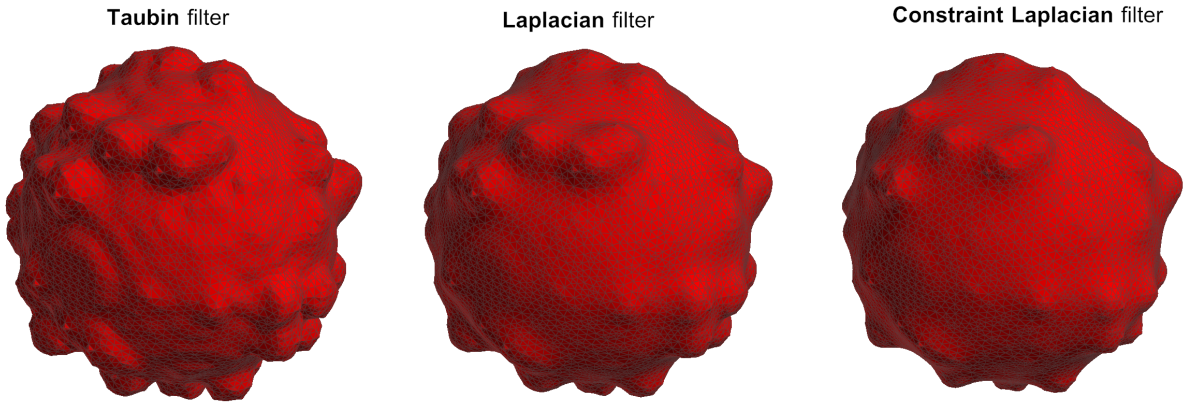


The **Taubin** filter iteratively combines different Laplacian filters to provide better volume preservation. The runtime for **Taubin** is longer than for Laplacian. In the images below, observe that the mesh is rougher than the result after **Laplacian** filtering, while its volume is closer to the volume of the original voxel structure.

Find more information on the Taubin filter in the [References](55).

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.

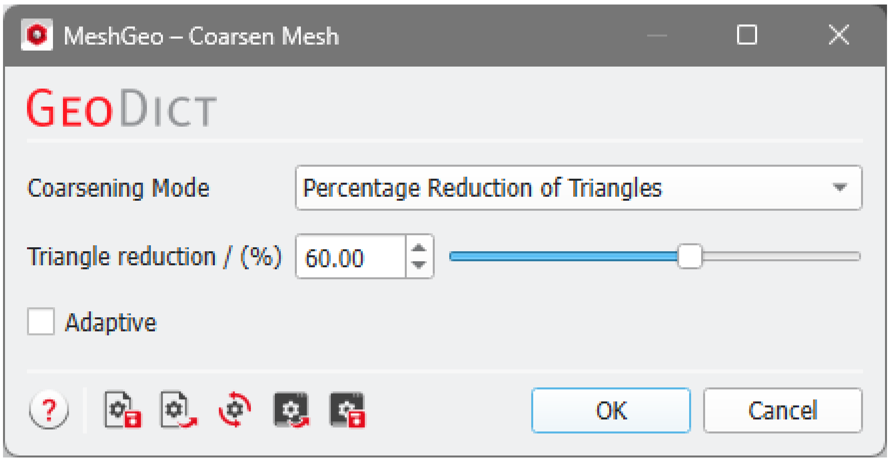


1.4.2 Coarsen Mesh

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 Mesh** dialog.

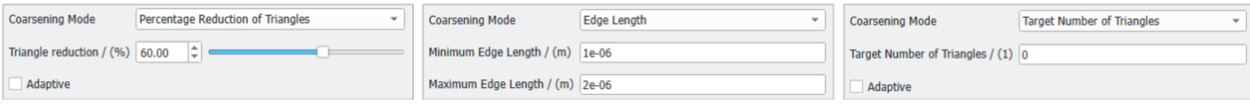
Generally, it is not supported to use Coarsening on the voxel surface mesh (**Create Mesh → Create Voxel Mesh → Voxel Surface**). Use the other voxel mesh modes, or smooth the voxel mesh before coarsening.



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. Depending on the choice of **Coarsening Mode**, the different stopping criteria and their corresponding input parameters are shown.

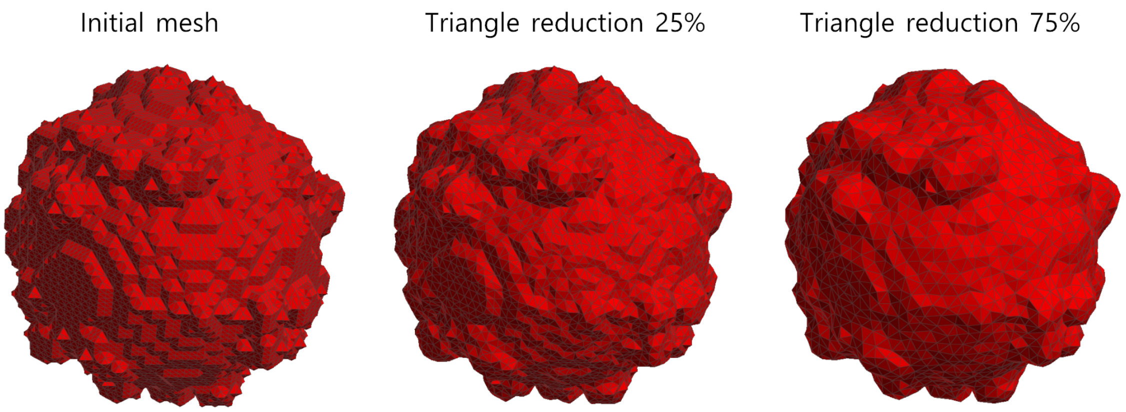


Percentage Reduction of Triangles

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

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

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



Edge Length

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

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

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

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

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

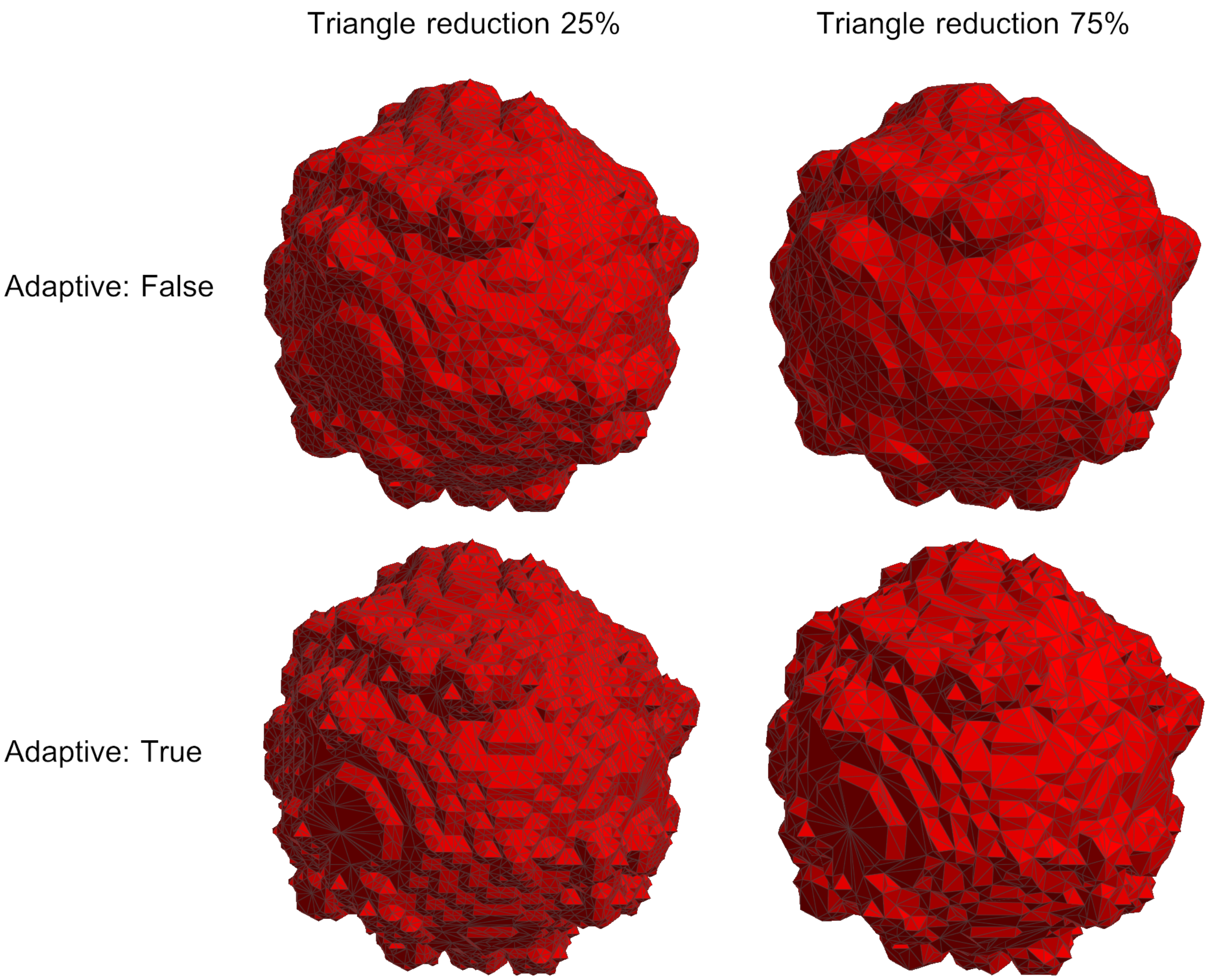
Target Number of Triangles

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

Adaptive

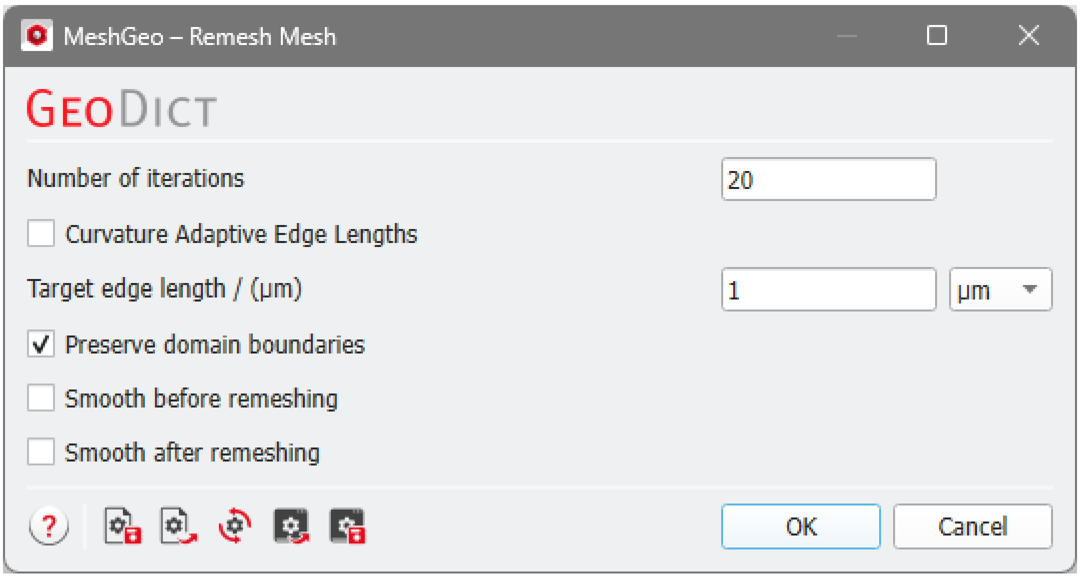
Adaptive is only available for the modes **Percentage Reduction of Triangles** and **Target Number of Triangles**. With **Adaptive**, the coarsening algorithm minimizes the deviation of the coarsened surface from the original surface. The **Adaptive** algorithm generates a mesh with a non-uniform triangle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

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



1.4.3 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.

Target 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 boundary will remain on the boundary after remeshing.

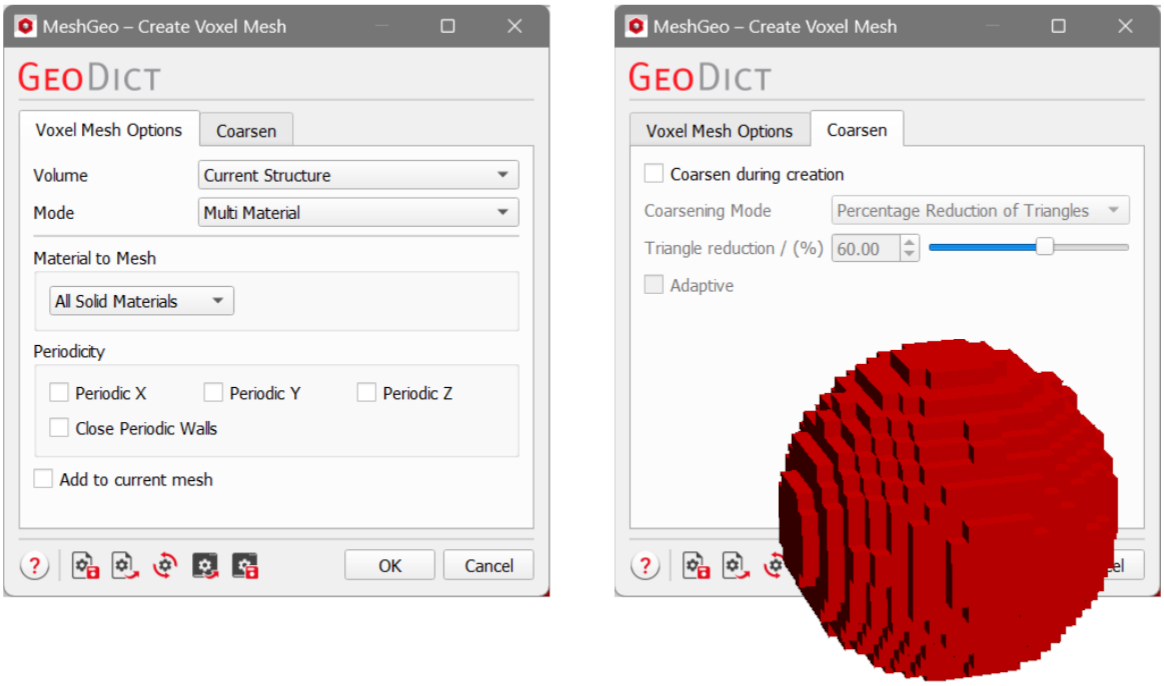
Smooth before remeshing

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. The **Taubin** filter is used in this option, as well as in **Smooth after remeshing.**

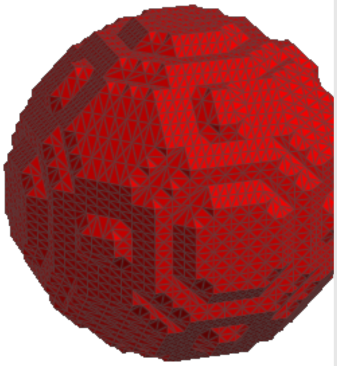
Smooth after remeshing

When **Smooth after remeshing** is chosen, the smoothing step is performed 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.

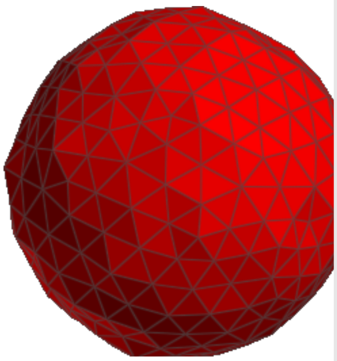
To illustrate the features, an example of a ball with 20 µm diameter with the voxel length 1 µm is used for meshing (**Create Mesh**), remeshing (**Remesh Mesh**), and smoothing (**Smooth Mesh**) to show the different meshes created.

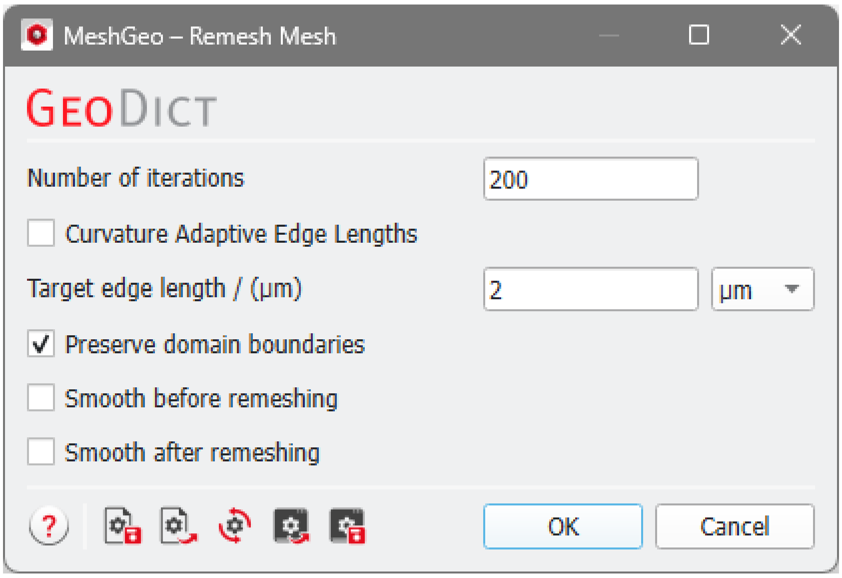


First, a voxel mesh is created with the settings shown in the dialogs above, without any **Coarsen**.

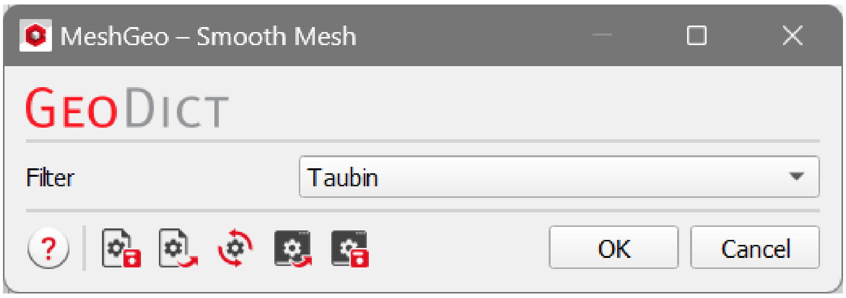


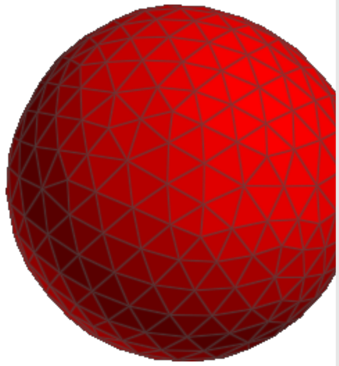
Remesh Mesh is executed with 200 as the **Number of iterations** and 2 µm as the **Target edge length**.



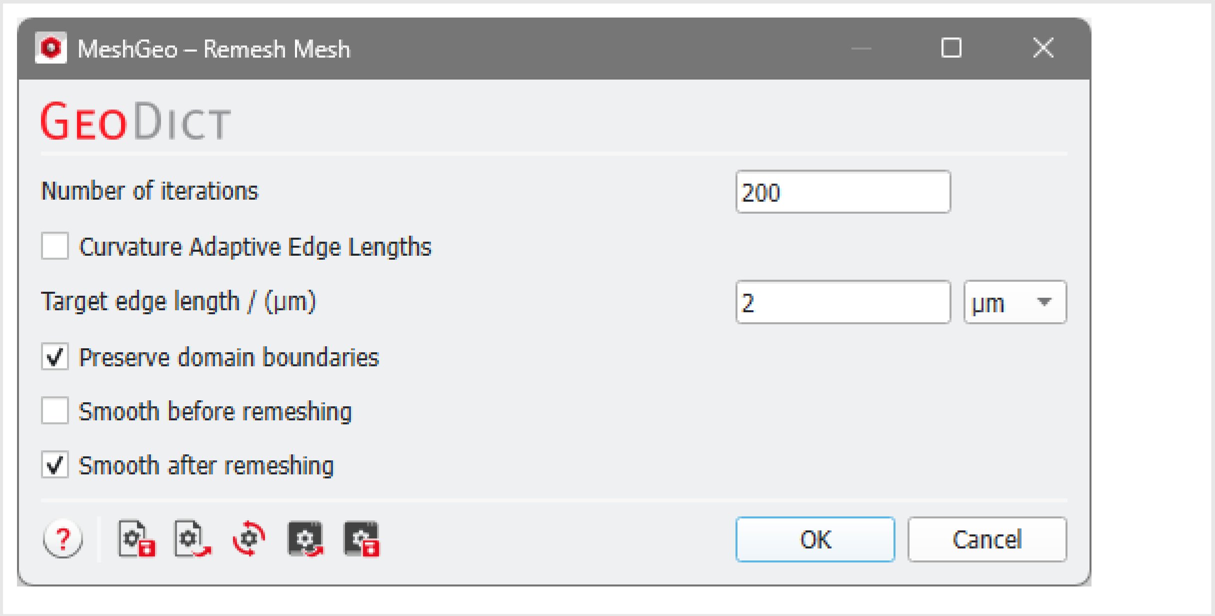


The mesh is then smoothed with the **Taubin** filter





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.

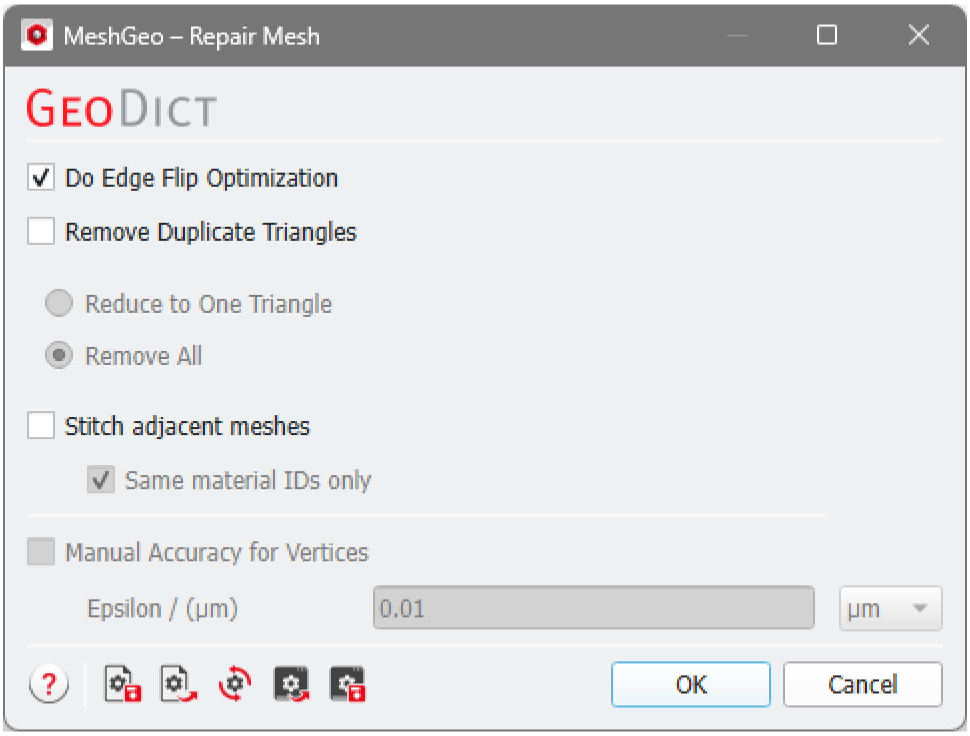

MeshGeo – Remesh Mesh
GeoDict
Number of iterations
200
Curvature Adaptive Edge Lengths
Target edge length / (µm)
2
µm
Preserve domain boundaries
Smooth before remeshing
Smooth after remeshing
OK
Cancel

1.5 Repair Mesh

Repair Mesh provides options to repair meshes.

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.

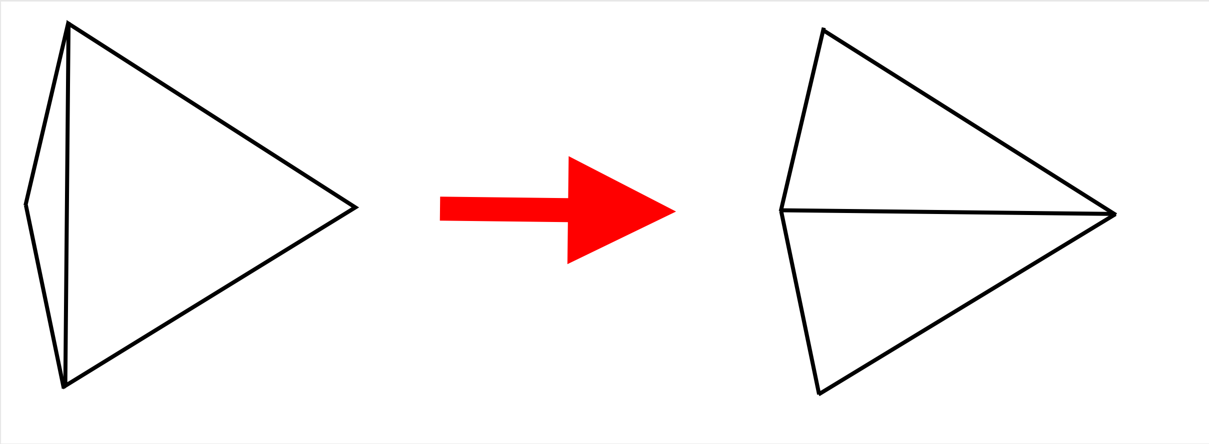


When all settings have been chosen, click **Repair** at the bottom of the **MeshGeo** section to create the mesh.

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.

Important! Choosing the option **Remove All** can lead to holes in the surface.

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

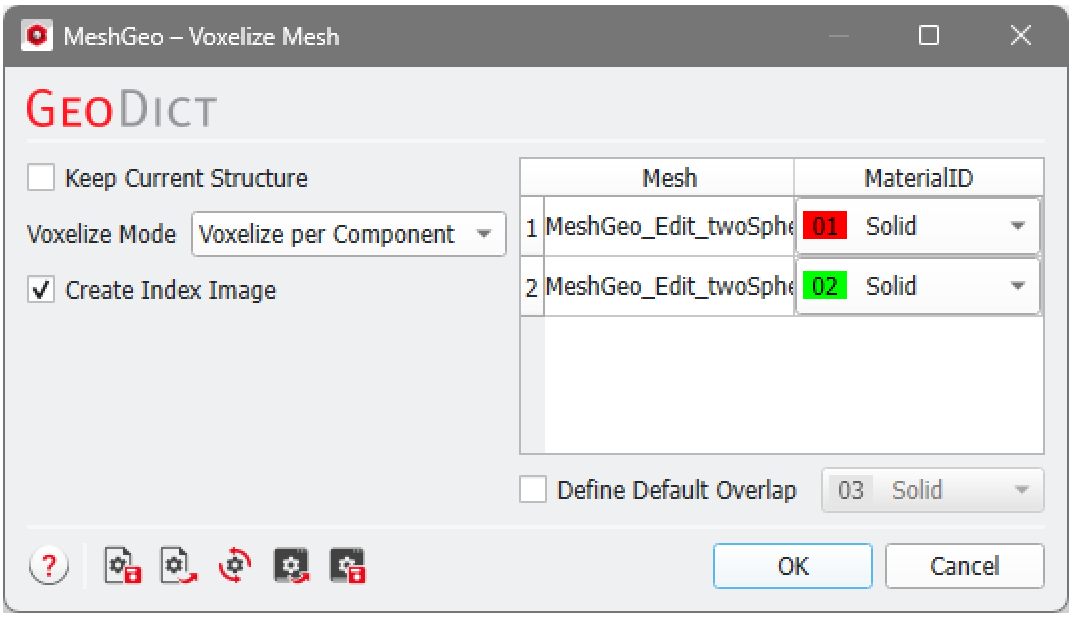
The option of **Manual Accuracy for Vertices** only takes effect when **Remove Duplicate Triangles** or **Stitch adjacent meshes** is used. It is to define a threshold to compensate for numerical errors in vertex coordinates. The mesh vertices that have a distance no more than the given **Epsilon / (µm)** are considered equal.

1.6 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 voxel structure.

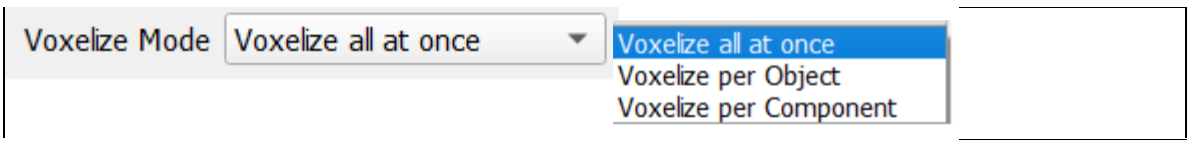


Some of the settings in the **Voxelize Mesh** dialog are analogous to the ImportGeo-CAD dialog. The more detailed explanation of the settings can be found in the ImportGeo-CAD chapter.

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** converts all loaded meshes to one Material ID in the voxel geometry.

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.

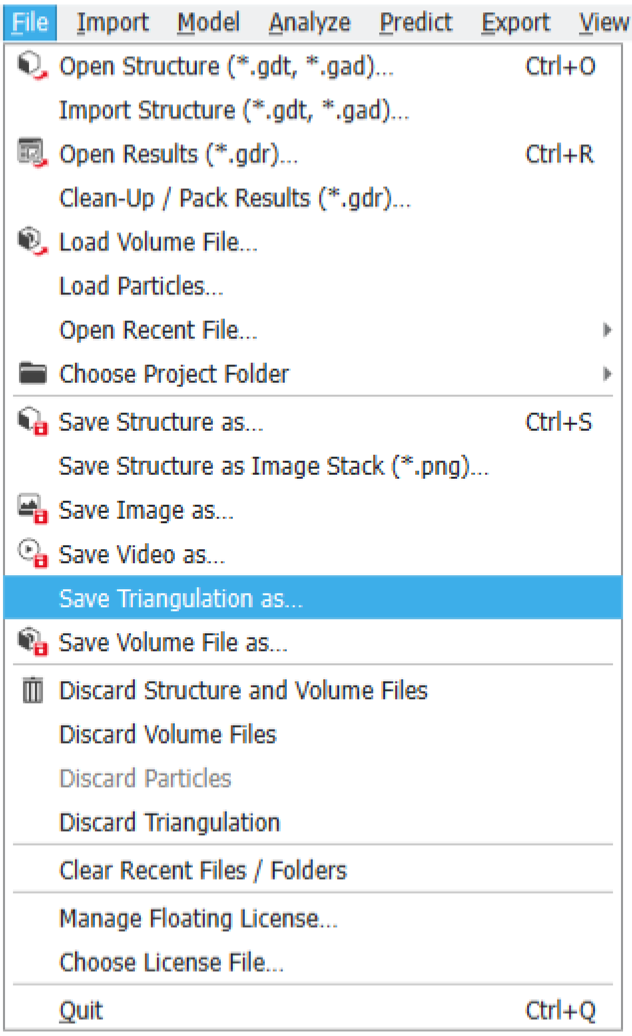
Define Default Overlap

With this option, you can assign specific materials 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.

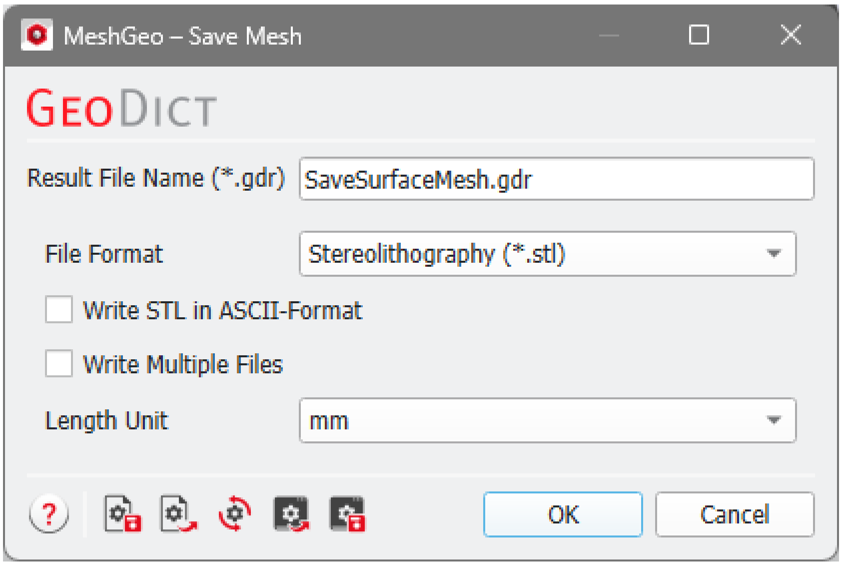
1.7 Save Mesh

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

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



A dialog opens which allows to select the settings for the saved file.



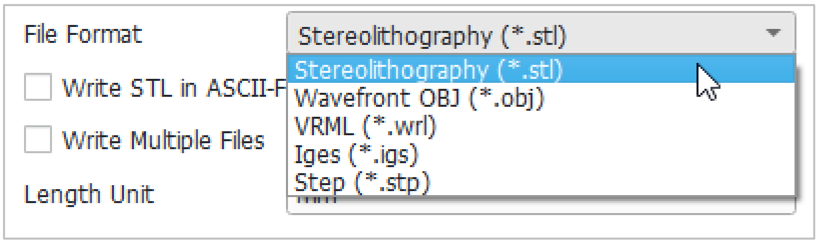
Result File Name

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

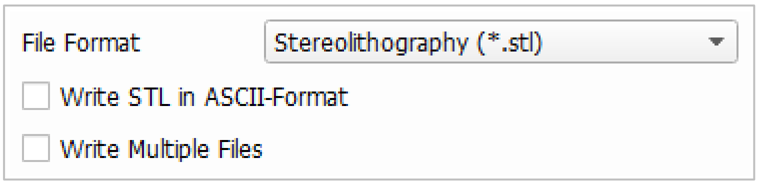
Select the **File Format** for the saved mesh.

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



Write STL in ASCII-Format

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

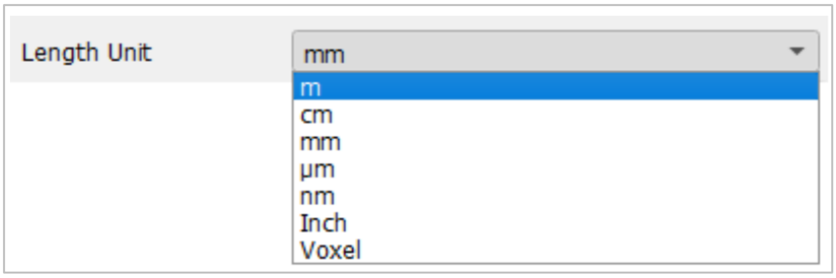


Write Multiple Files

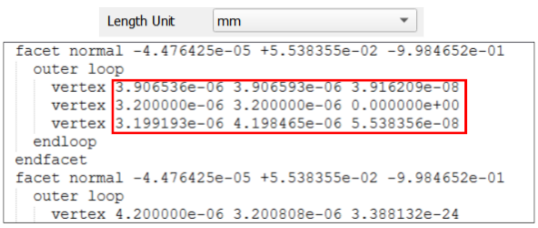
This option is also only available when the *.stl file format is chosen. When the STL format is chosen and multiple meshes are currently loaded, these meshes can be saved to different files if necessary. This option is enabled with **Write Multiple Files**. For the other file formats, multiple meshes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

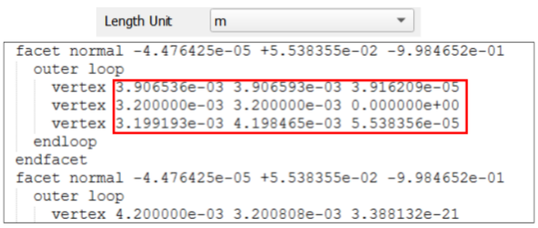
Length Unit

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



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





1.8 Export as Mesh

The **Export as Mesh** commands in MeshGeo havethe same functionality as those in ExportGeo-CAD. Instead of having the mesh loaded in GeoDict, the **Export as Mesh** commands save the created mesh directly to the hard disk. Therefore, visualization and editing of the meshes after creation are not possible.

The parameters of **Export as Mesh** includes both mesh creation 5 and mesh saving 51. They are all described in the respective chapters of the ExportGeo-CAD User Guide:

- **Export Voxel Mesh**
- **Export Object Mesh**
- **Export Object CAD**

1.9 References

G. Taubin, *Curve and surface smoothing without shrinkage*, Proceedings of IEEE international conference on computer vision (1995), https://doi.org/10.1109/ICCV.1995.466848

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

MATH
2 MARKET

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