ExportGeo-CAD / MeshGeo

User Guide

GeoDict release 2022

Published: November 25, 2021

GeoDict

https://doi.org/10.30423/userguide.geodict2022-exportgeocad

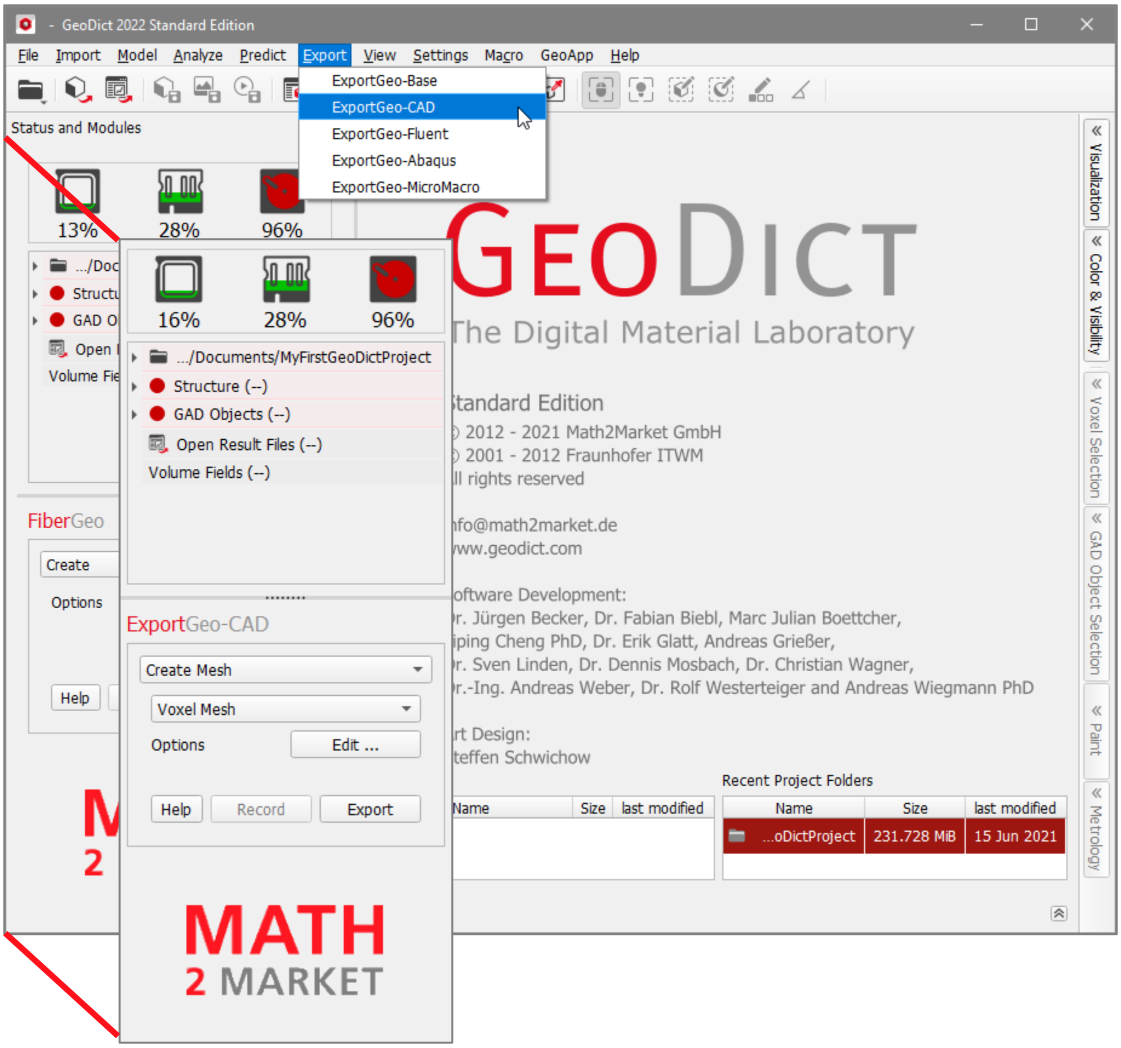
Convert 3D structures to surface meshes – MeshGeo / ExportGeo-CAD

With ExportGeo-CAD and MeshGeo, 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 to transfer the object information from GeoDict's structure generators to external software.

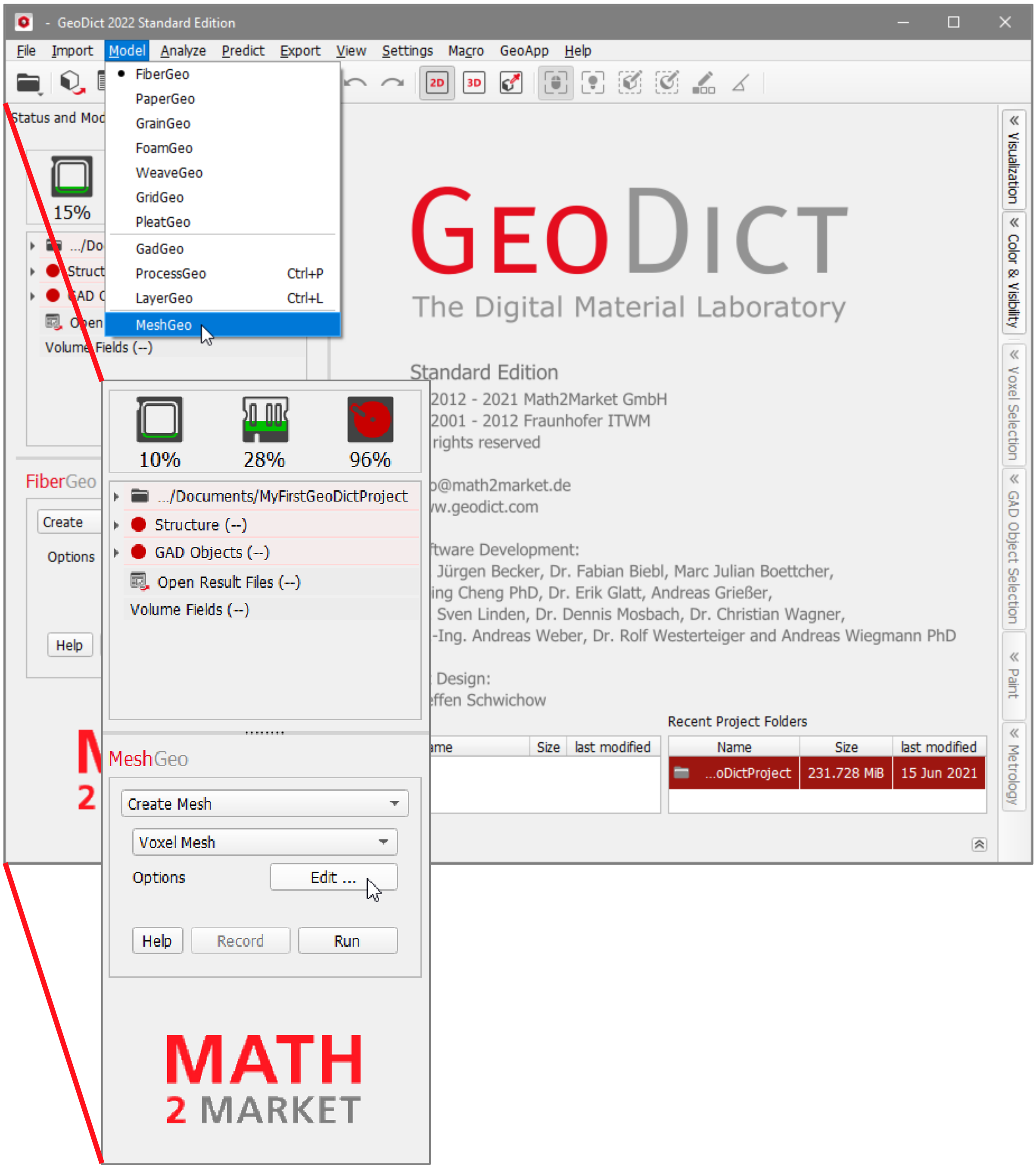
- ExportGeo-CAD saves the created mesh directly to the hard disk. Therefore, it is not possible to edit the mesh after creation.
- MeshGeo keeps the mesh in memory after creation and allows subsequent smoothing and coarsening operations. For this, MeshGeo contains the additional options **Smooth Mesh**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

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



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



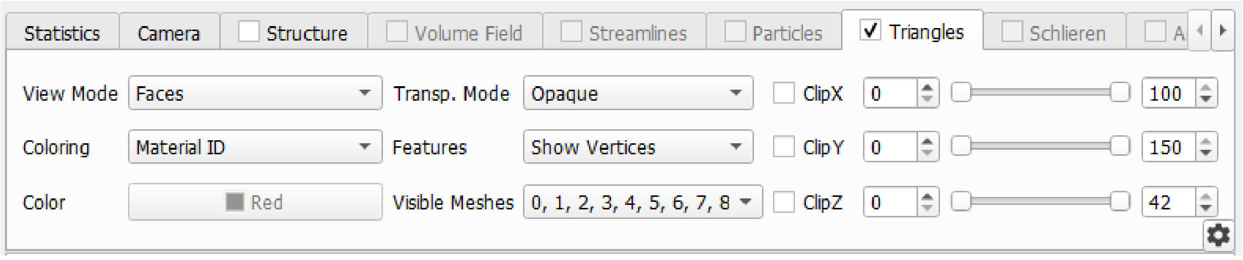
In the **ExportGeo-CAD** or **MeshGeo** section, to the left of the Visualization area, a pull-down menu displays the choices for the meshing options. The options available for **Create Mesh** are the same for both modules.

Additionally, MeshGeo contains options to further edit and save the mesh. For ExportGeo-CAD, the mesh is directly saved after the creation. The available options to save the mesh are the same for both modules and explained together in pages 25 ff.

VISUALIZE SURFACE MESHES IN GEODICT

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

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



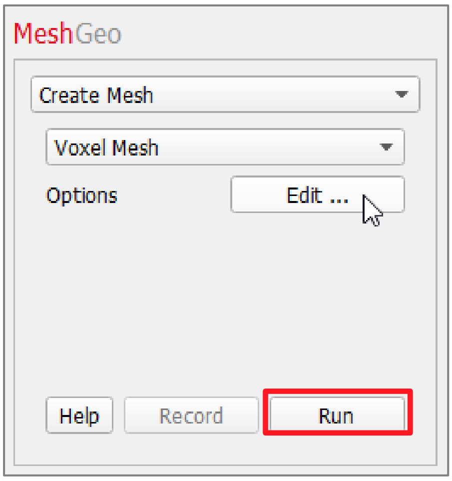
Create Mesh

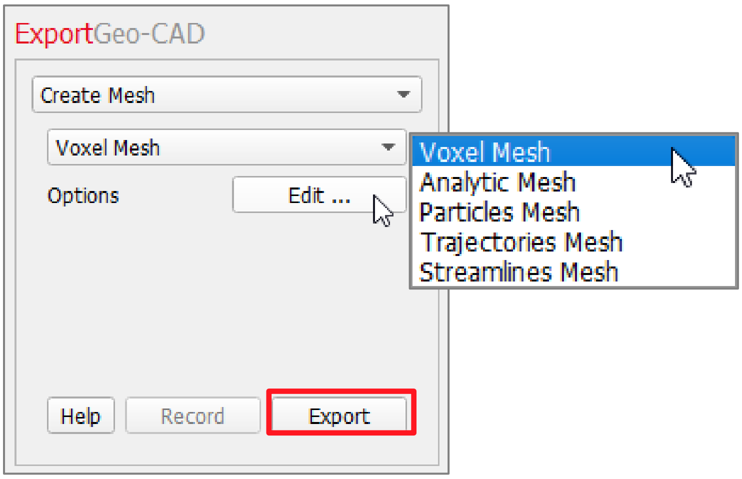
Select **Create Mesh** in MeshGeo or ExportGeo-CAD and specify the type of mesh from the pull-down menu. Meshes of a structure in GeoDict can be created based on the voxel information (**Voxel Mesh**). This voxel information can be a structure, a grey value image, an index image, or a volume field.

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

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

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





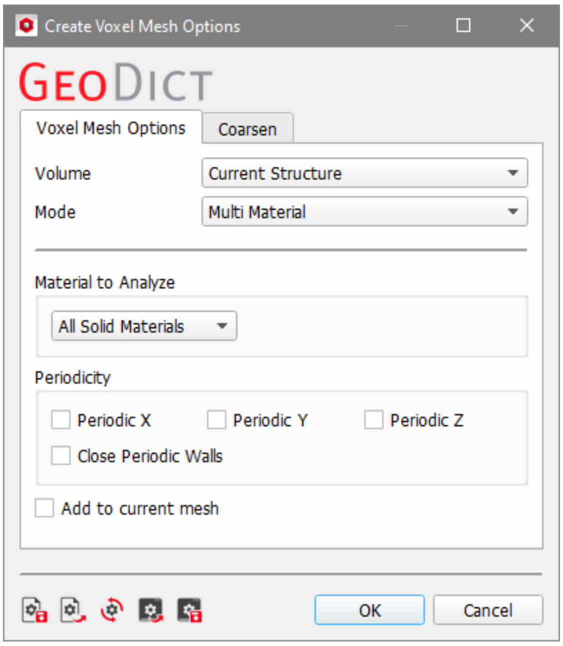
When all settings have been chosen, click **Run** at the bottom of the **MeshGeo** or **Export** at the bottom of the **ExportGeo-CAD** section to create the mesh.

Voxel Mesh

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

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

It contains the same options as **Save Mesh** in MeshGeo. See page 25 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 choice for **Volume** and **Mode** change the other available options.

Volume

In GeoDict 2021, the option to select the **Volume** was introduced in **Create Mesh → Voxel Mesh**. The available options are explained below.

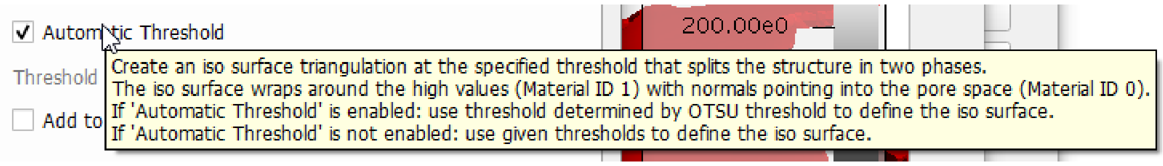
Current Structure

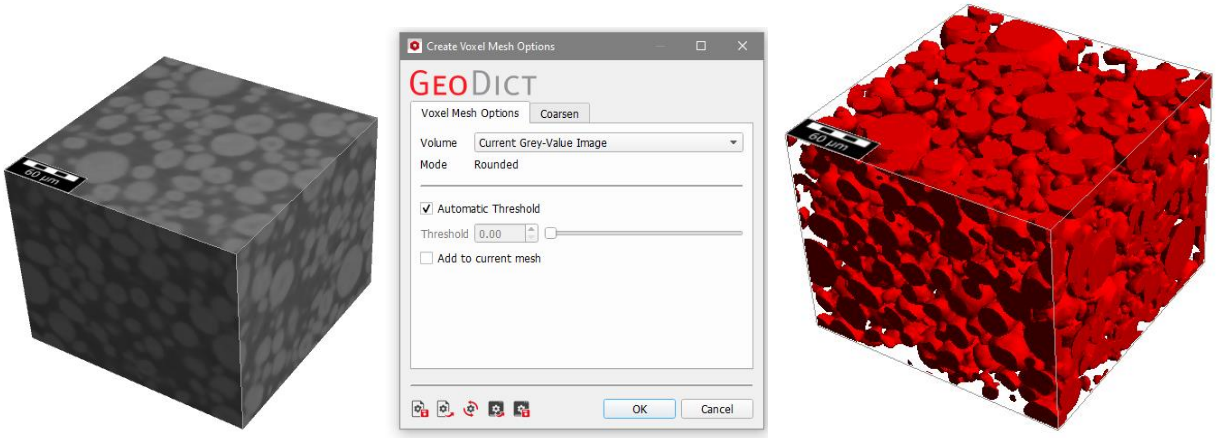
Meshing the **Current Structure** was the only available option prior to GeoDict 2021. With this option, the currently loaded voxel structure is converted to a mesh. For Current Structure, all four **Mode**s are available: **Voxel Surface**, **Rounded**, **Smooth** and **Multi Material**.

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 directly mesh the **Current Grey-Value Image**. In this case, the **Mode** is fixed to **Rounded**. Based on the given **Threshold**, an iso surface is created that splits the structure in two phases. The normal of the iso surface point into the pore space. If **Automatic Threshold** is selected, the OTSU algorithm is used to determine the threshold.

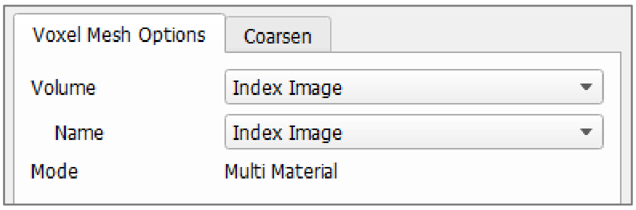
With this option, all information from the grey value image can be used. The created mesh is smoother than a mesh created from a segmented voxel structure, and the additional grey value information allows to obtain sub-voxel precision.



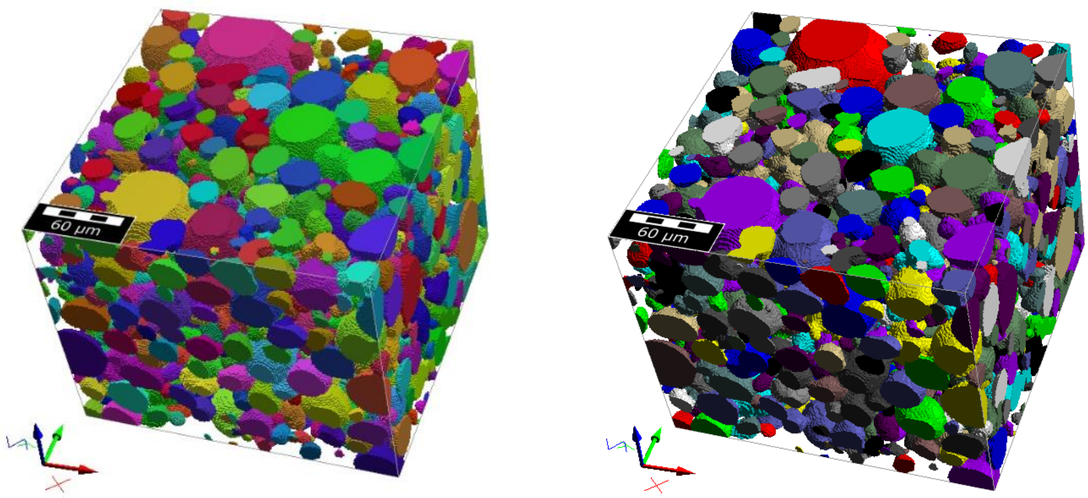


Index Image

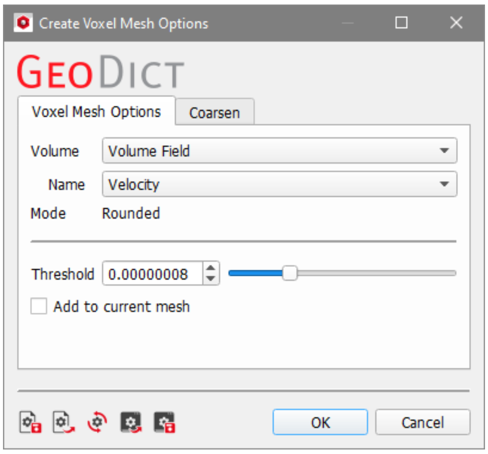
By choosing an **Index Image** as Volume, it is possible to use the object indexes to create **Multi Material** triangulations. Analogously, the object indexes from GrainFind or FiberFind results can be used in the same way. Each object with a given index is converted to its own mesh, and the meshes for different objects do not overlap. The **Mode** is fixed to **Multi Material**, since this is the option that keeps the object information and preserves the partition of unity.



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

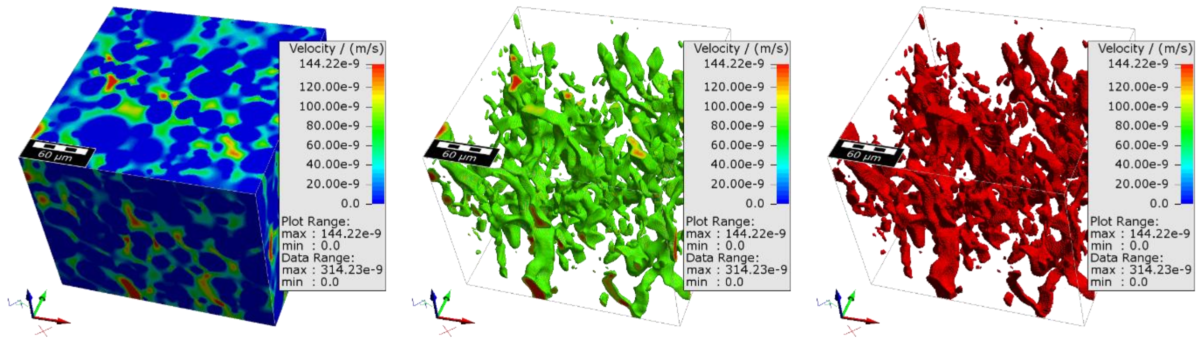


Volume Field



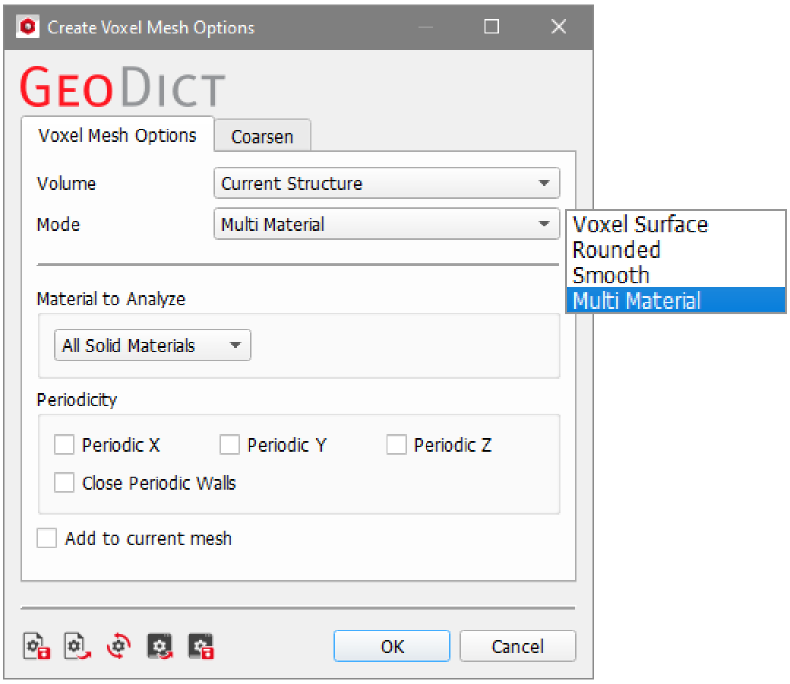
With the Volume Field option, triangulations can be created from arbitrary GeoDict results. In the example below, a flow field from a FlowDict simulation is converted to a mesh. The **Volume Field** option works analogously to the **Current Grey-Value Image** option: It creates an iso-surface mesh based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

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



Mode

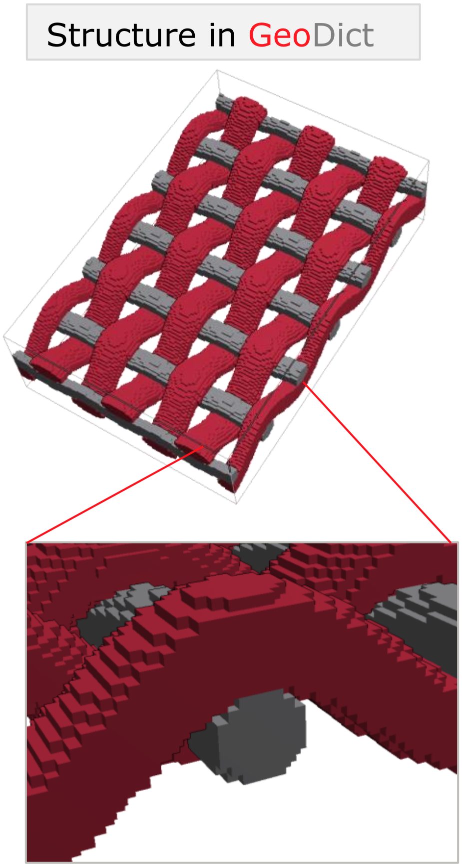
Four **Mode**s can be used to export voxel data: **Voxel Surface**, **Rounded**, **Smooth**, or **Multi Material**. **Multi Material** is the default. Which options are available depends on the selected Volume: For example, if the **Current Grey-Value Image** is selected, only **Rounded** is available.

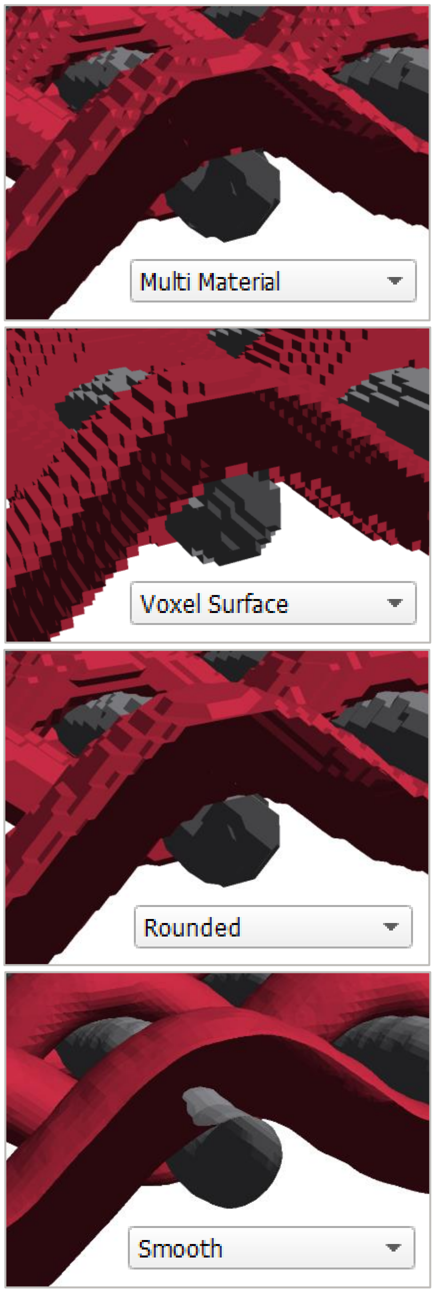


Voxel Surface exports the exact surface of the voxel structure with its characteristic rectangular edges. This option is available in GeoDict to ensure compatibility with older releases. In general, it is not recommended to use **Voxel Surface** since the **Multi-Material** mode leads to better results.

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

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





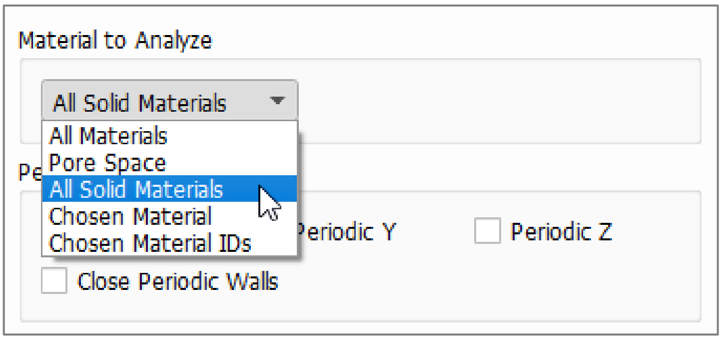
With the **Multi Material** option, the meshes for the different materials do not overlap. They have perfect contact at the points where different materials touch. When all materials are exported, a partition of unity is created. This means that each point in the domain is enclosed in exactly one surface.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

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

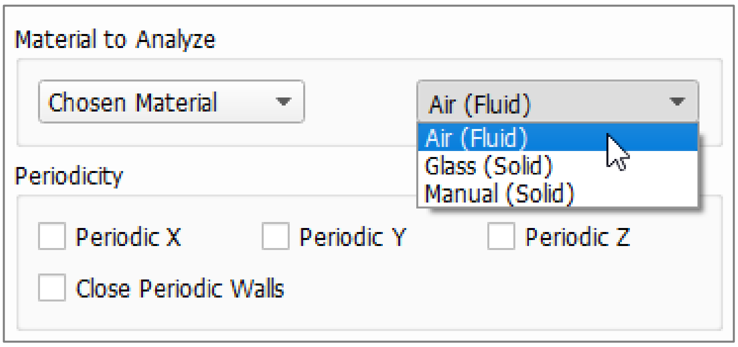
Material to Analyze

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

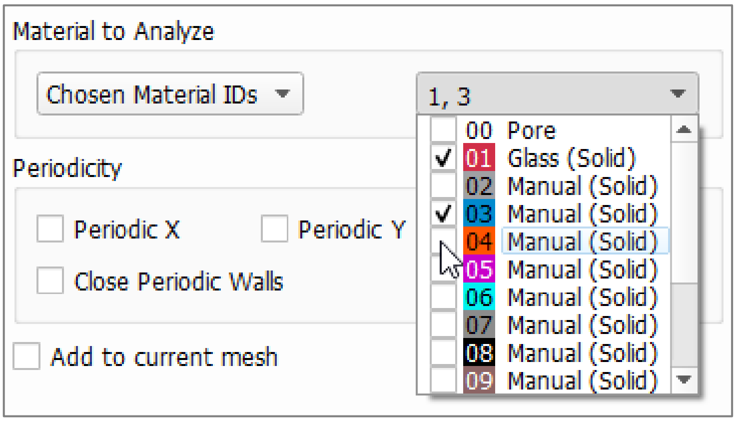
All Solid Materials is the default, since this is the best choice for most use cases.



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



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



Periodicity

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

Close Periodic Walls

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

Avoid Repeating Edges

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

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

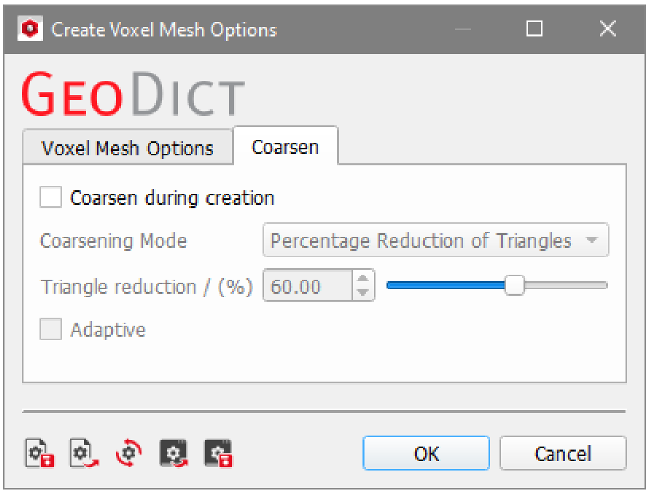
Add to current mesh

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

See Smooth Mesh in page 19 and Coarsen Mesh in page 22

Coarsen

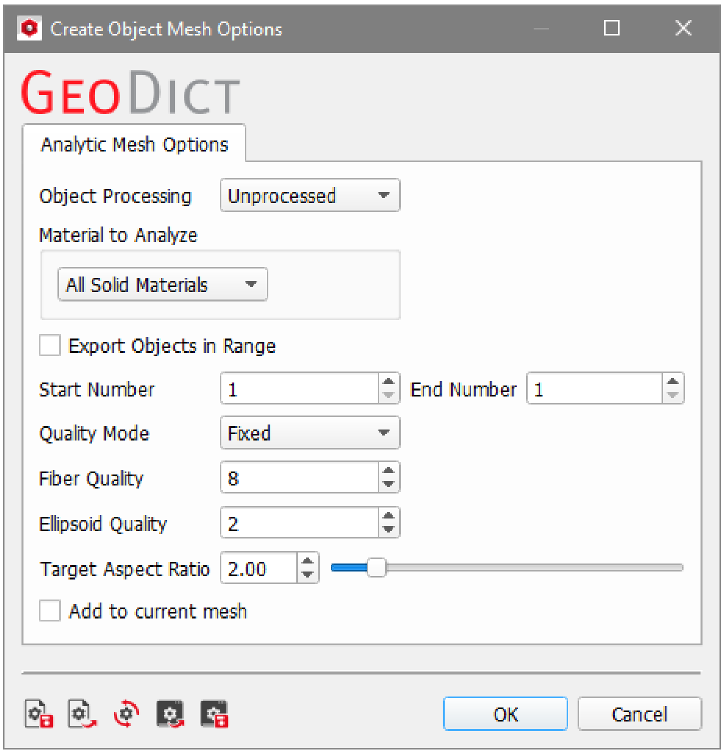
Coarsening can either be applied during mesh creation or afterwards. The settings are the same, but the results may differ – especially if the **Adaptive** mode is used. For further information, see page 22.



Analytic Mesh

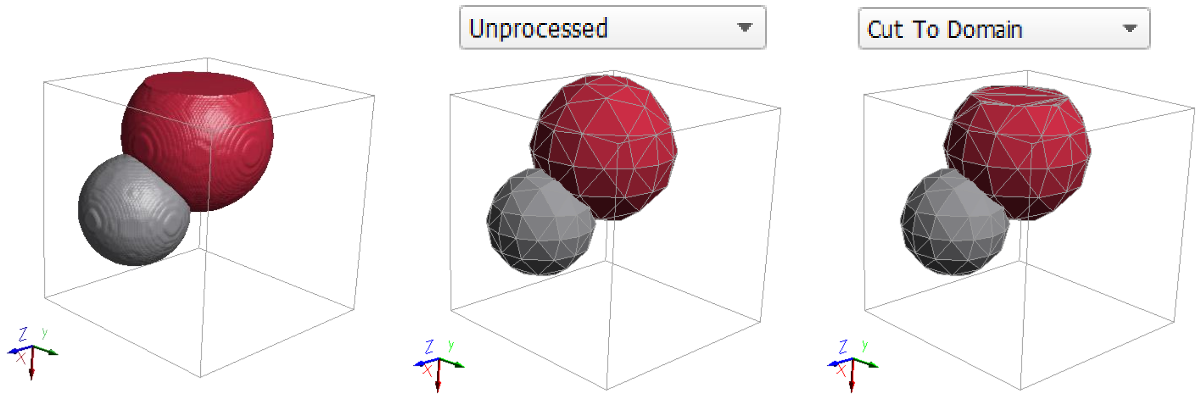
To export the objects in the structure into a mesh, select **Analytic Mesh** from the pull-down menu. Objects in GeoDict can overlap. With **Analytic Mesh**, each object is meshed separately, therefore the meshes for the different objects can overlap, too. If it is important that these meshes do not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

Analytic Mesh does not work for combined objects or intersected objects. Check out the GAD Format handbook of this user guide for more information about these object types.



Object Processing

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

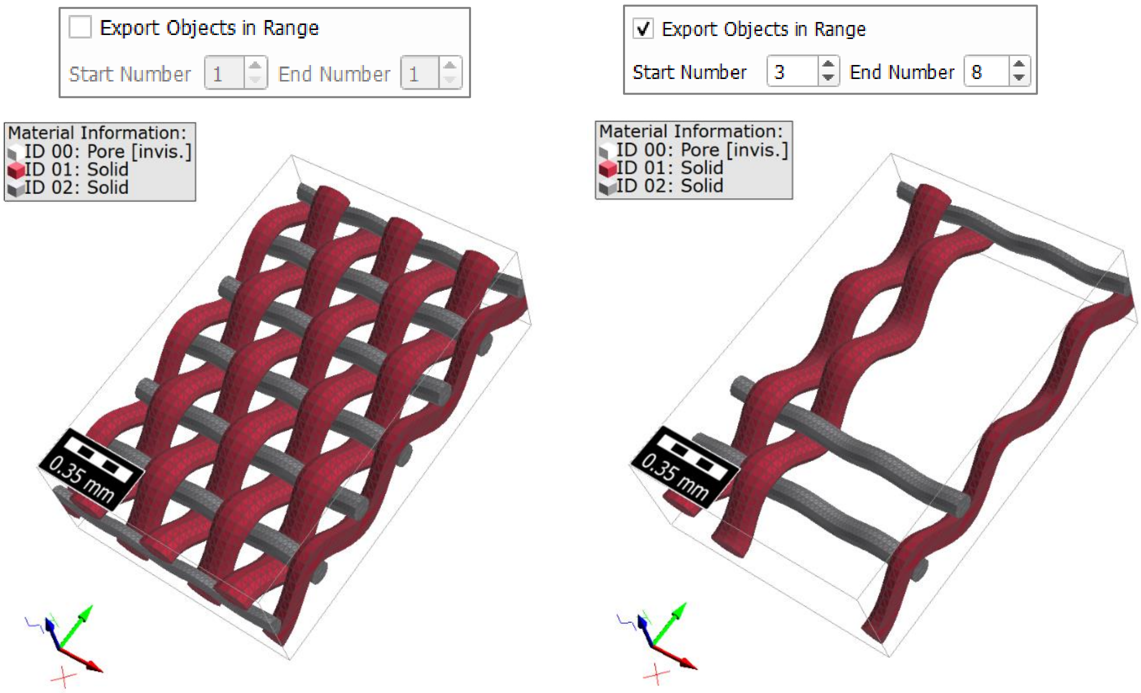


Material to Analyze

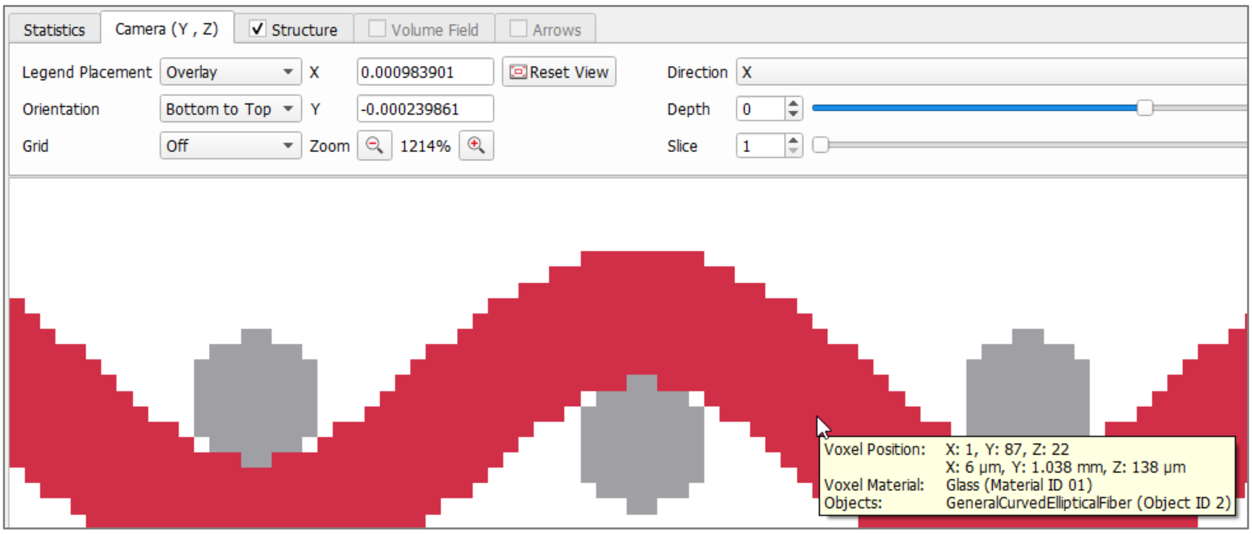
The choices for **Material to Analyze** are explained on page 9. For **Analytic Mesh**, not all material IDs in the structure can be used to create a mesh. For example, the pore space is not an object in GeoDict and therefore no analytic mesh can be created from it. Furthermore, the overlap between objects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

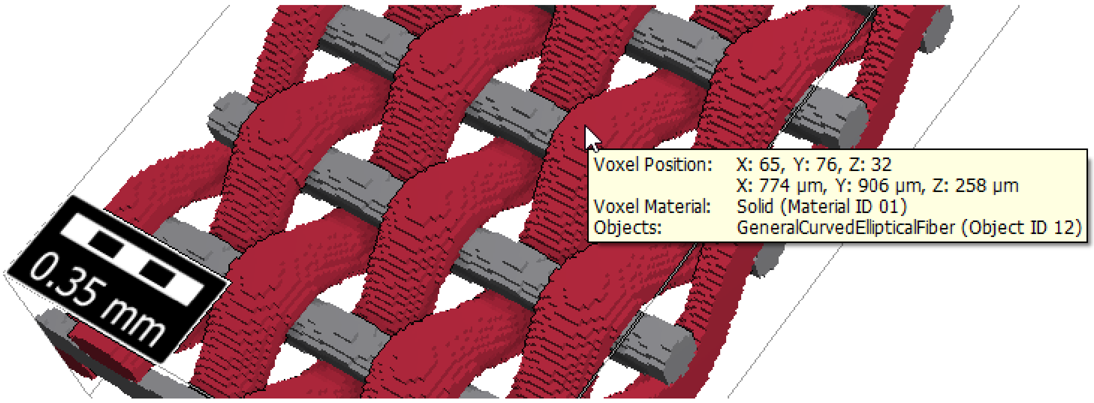
Export Objects in Range

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



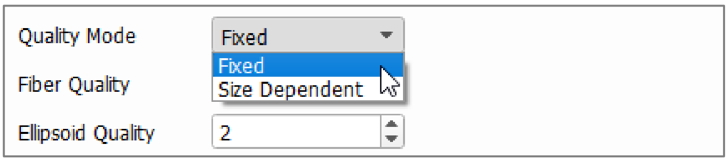
To find the object number of a given object, hover the mouse over the object in 2D or 3D view and check out the ToolTip.





Quality Mode

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



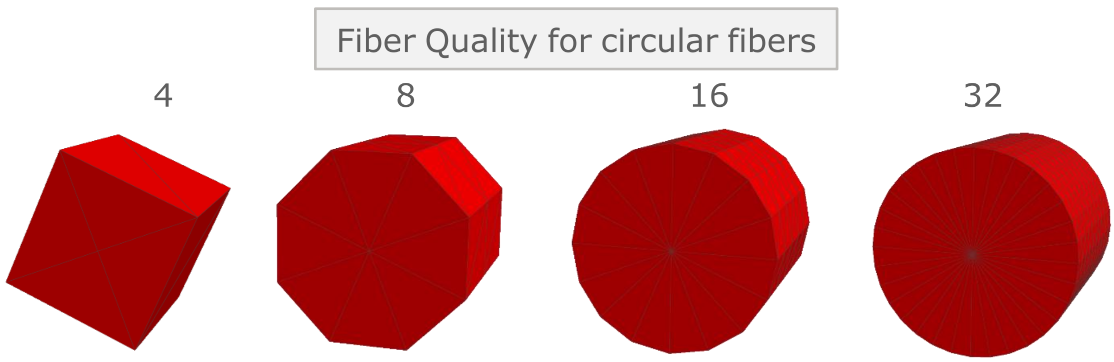
With **Quality Mode** set to **Fixed**, the **Fiber Quality** and **Ellipsoid Quality** are defined globally for the structure. When choosing **Size Dependent**, minimal and maximal values must be defined. This option is recommended if the size of the objects in the structure varies by a large amount.

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

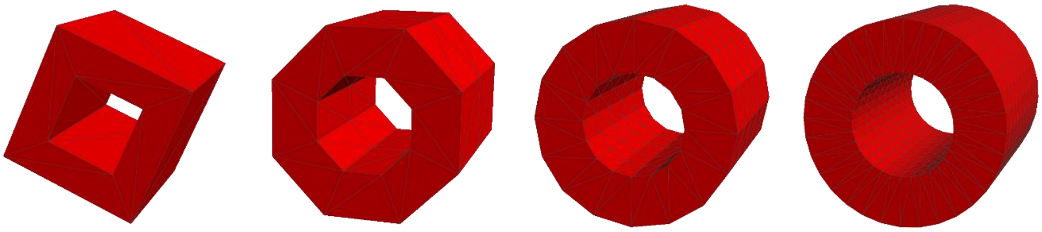
Fiber Quality

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

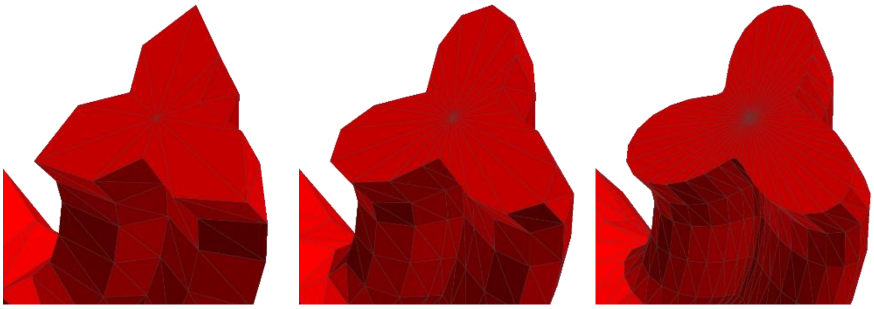
- For fibers with circular and ellipsoidal cross-section, the **Fiber Quality** corresponds to the number of nodes with which the cross-section of the fiber is discretized. This does also hold for fibers of hollow and cellulose types.
- For rectangular and angular fibers, the **Fiber Quality** has no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



Fiber Quality for hollow fibers



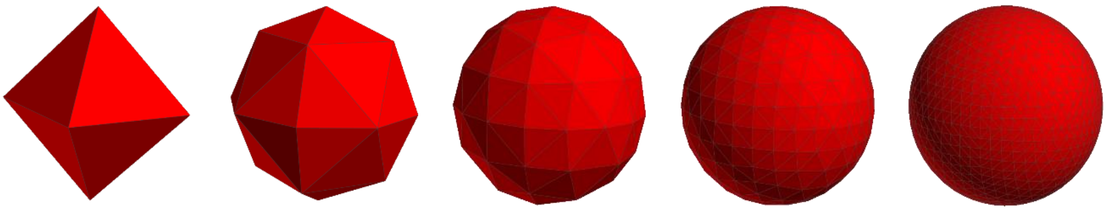
Fiber Quality for Rosetta fibers



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.

Ellipsoid Quality



Target Aspect Ratio

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

Add to Current Mesh

See Add to current mesh, in page 10.

Particles Mesh

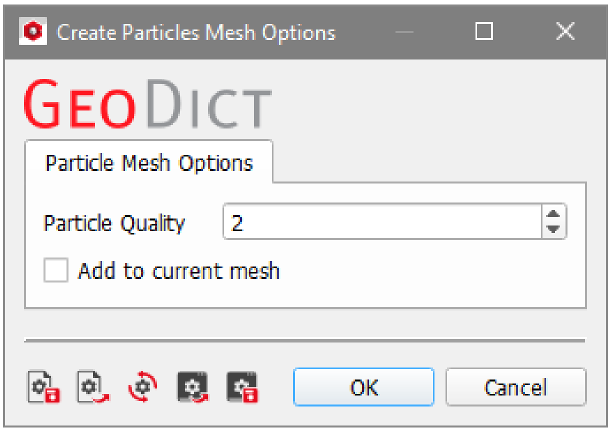
Particle positions and trajectories can be obtained from FilterDict and AddiDict simulations. MeshGeo can create a mesh from the currently loaded particles in GeoDict. The mesh is always created at the particle positions currently shown in GeoDict. Particles and trajectories can be loaded e.g. from a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

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

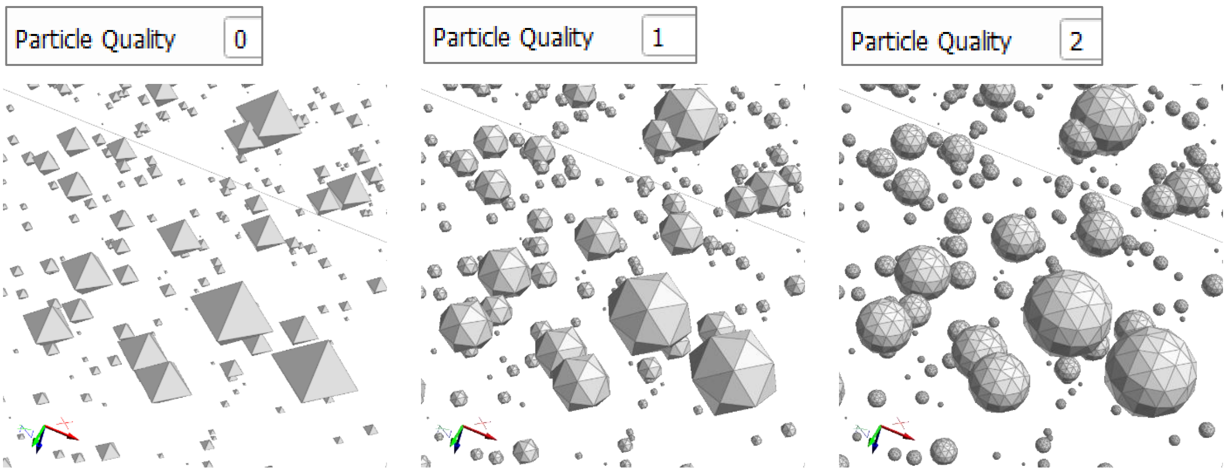
Start MeshGeo, select **Create Mesh** and **Particles Mesh** in the MeshGeo section and click the **Options**' **Edit...** button to set the parameters. When all settings are chosen, click the **Run** button at the bottom of the MeshGeo section to create the mesh.

Particle Quality

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



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



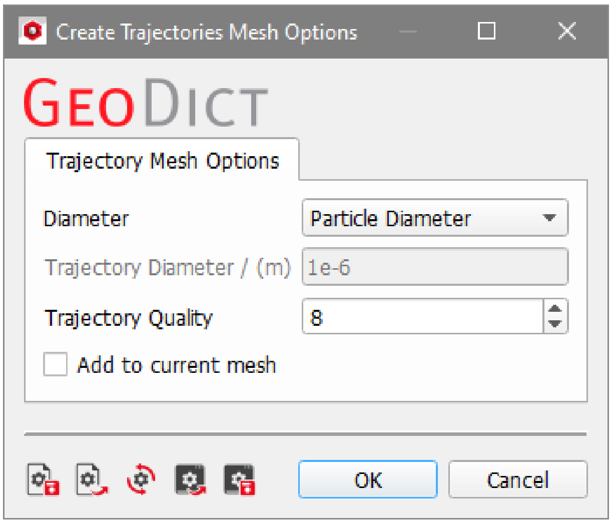
Trajectories Mesh

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

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

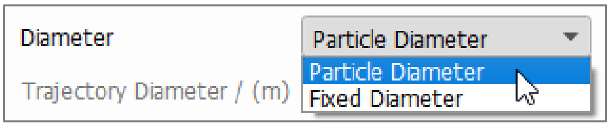
Under the **Particle Visualization** (FilterDict) or **Data Visualization** (AddiDict) tab, make sure that **Trajectories** is checked and click the **Load Particles** button.

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



Diameter

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

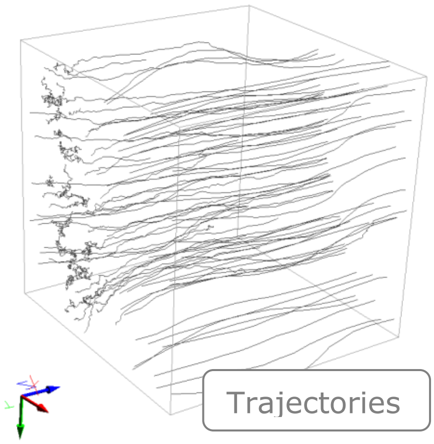


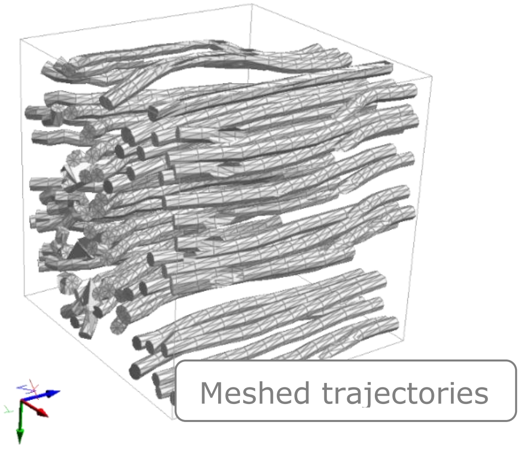
Trajectory Quality

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

Add to current Mesh

See Add to current mesh in page 10.

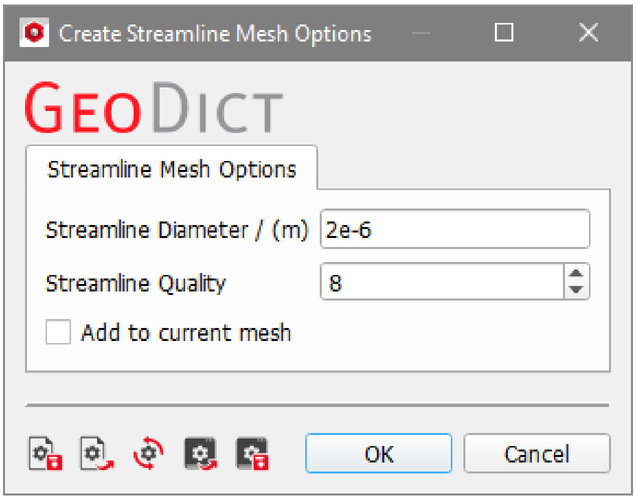

Trajectories


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



Streamline Diameter

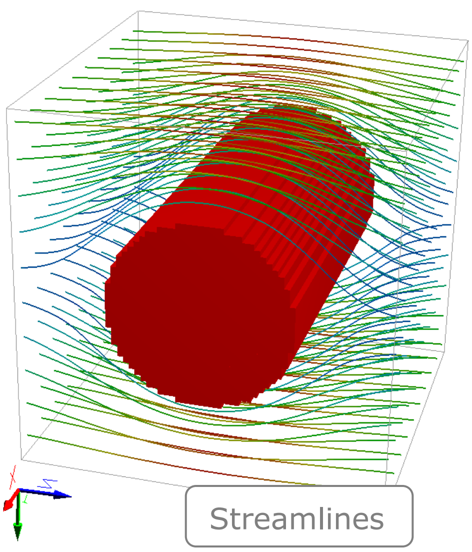
The **Streamline Diameter** option defines the diameter of the meshed streamlines.

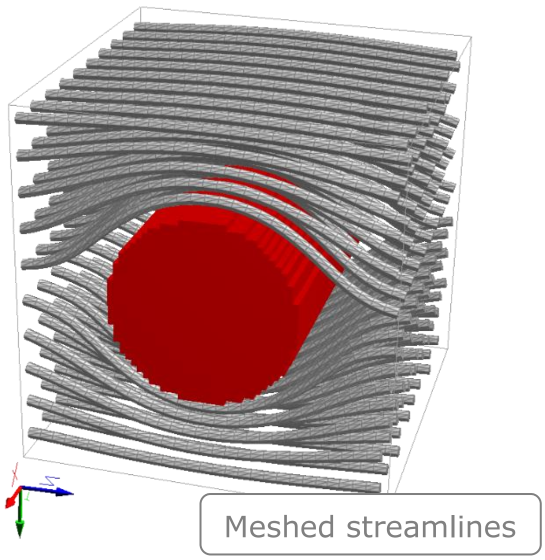
Streamline Quality

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

Add to current mesh

See Add to current mesh in page 10.

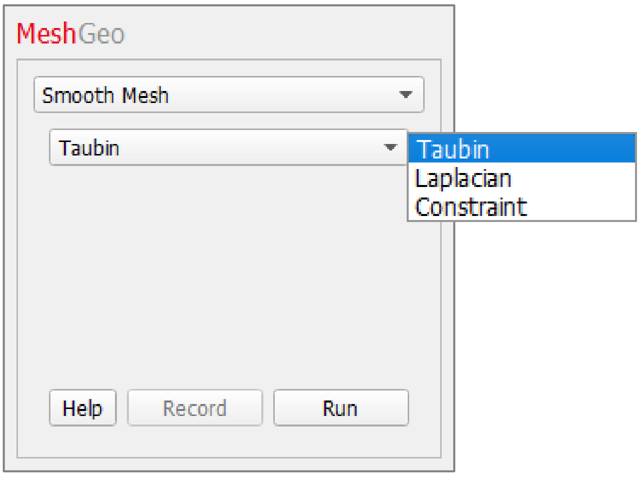




Smooth Mesh

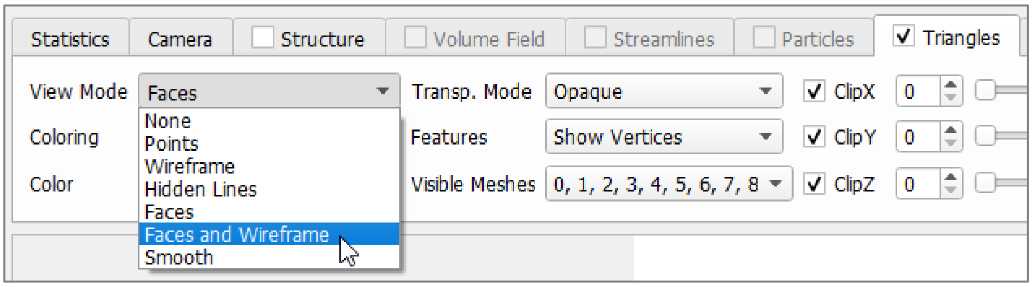
Meshes from voxel structures often have a significant surface roughness which can be reduced by applying smoothing filters.

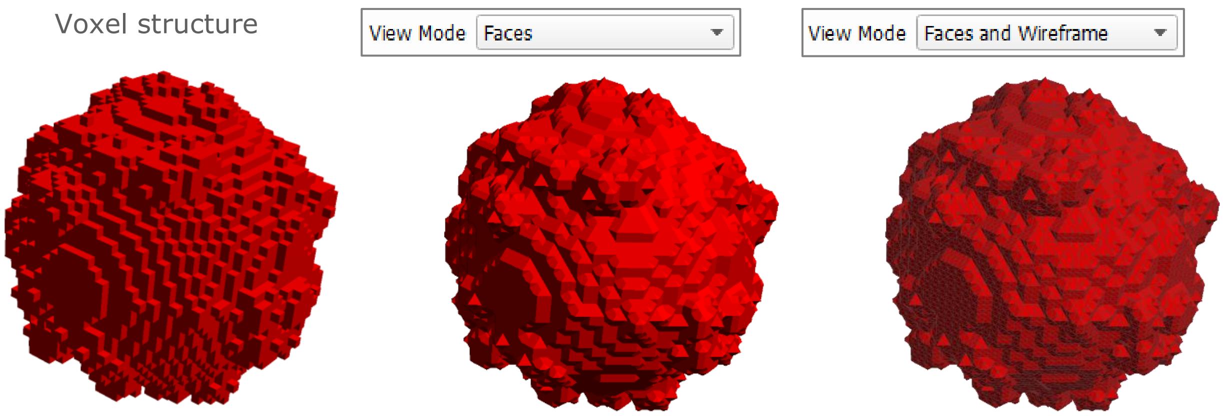
In MeshGeo, three smoothing filters are selectable from the pull-down menu to improve the mesh quality: **Taubin**, **Laplacian**, or **Constraint**. Choose a filter and click **Run** to apply the selected filter to the current mesh.



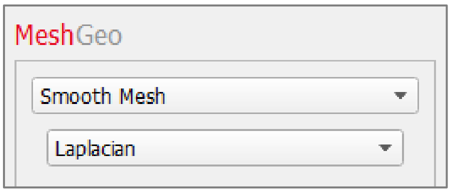
Smoothing is best shown on a structure with an irregular surface. The example structure created here is a grain with surface roughness. The original mesh before smoothing is created with **Create Mesh** → **Voxel Mesh** and the Multi Material mode (see page 9).

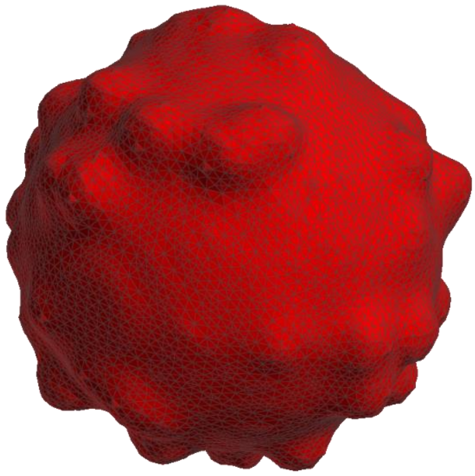
To observe the difference between the different mesh types, it is recommended to set the **View Mode** to **Faces and Wireframe** (**Triangles** tab → **View Mode** → **Faces and Wireframe**, in the Visualization panel above the Visualization area).





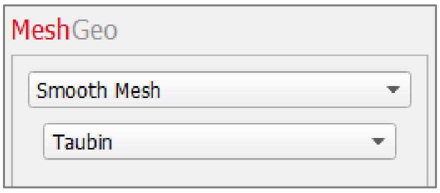
The **Laplacian** filter is a simple and fast option for mesh smoothing. However, this filter causes mesh shrinking for most geometries. For further information, see Wikipedia: Laplacian Smoothing.

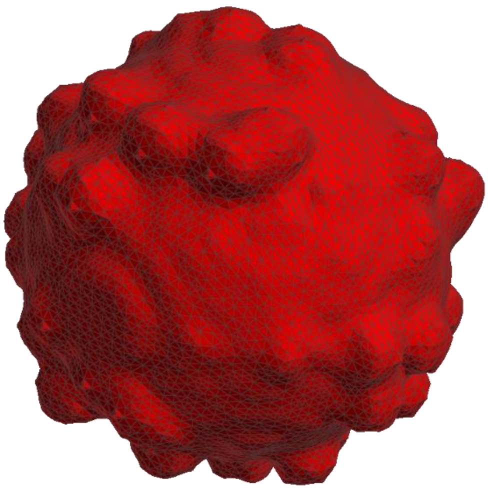




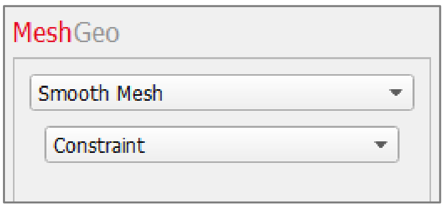
The **Taubin** filter is the default smoothing filter in GeoDict. This filter iteratively combines different Laplacian filters to provide a better volume preservation. The runtime for **Taubin** is longer than for Laplacian. In the screenshot on the right, observe that the mesh is rougher than the result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

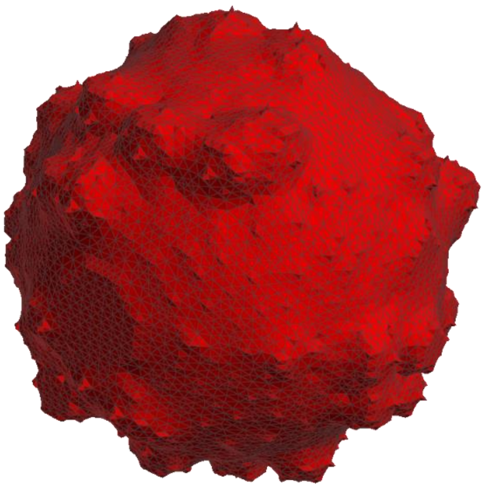
For further information about the Taubin filter, see [Reference (1)].





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





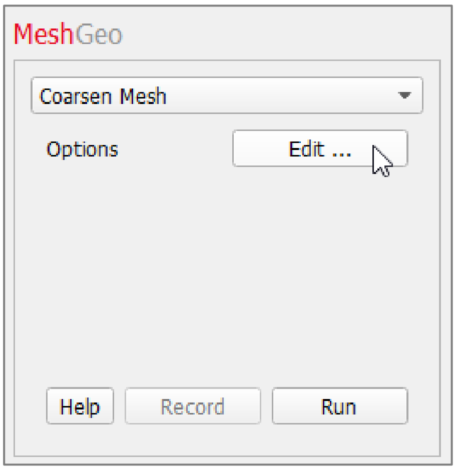
References

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

Coarsen Mesh

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

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

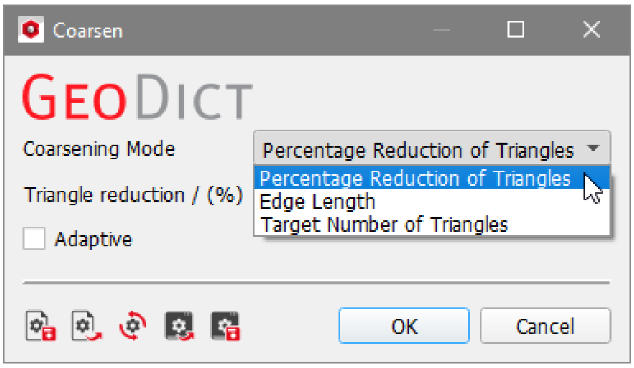


Generally, it is not supported to use Coarsening on the voxel surface mesh (**Create Mesh → Voxel Mesh → Voxel Surface**). Use the other voxel mesh modes or previous mesh smoothing instead.

Coarsening Mode

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

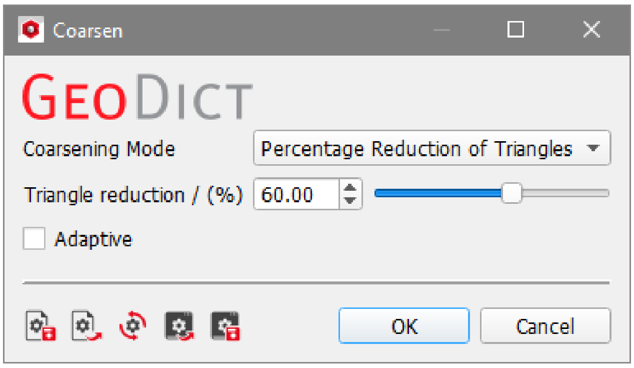
The coarsening algorithm reduces the number of triangles in the currently loaded mesh by combining short edges to longer edges.



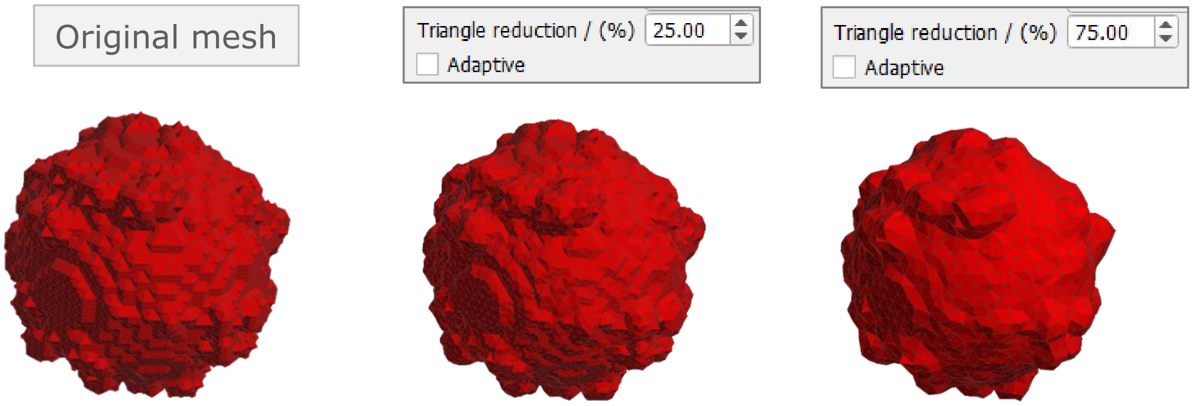
Percentage Reduction of Triangles

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

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



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

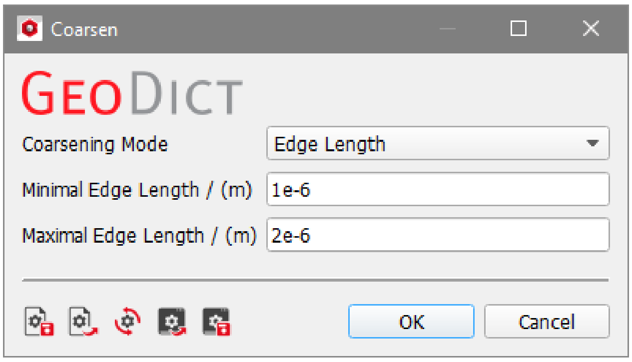


Edge Length

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

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

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

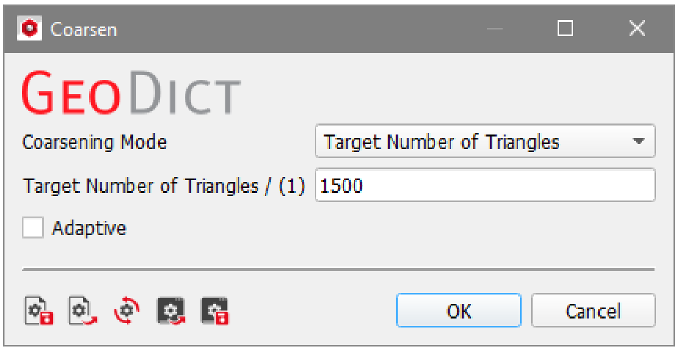


The coarsening algorithm takes all edges smaller than the minimal edge length and combines them until no edge is smaller than the minimal edge length. The algorithm tries not to exceed the maximal edge length in the process.

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

Target Number of Triangles

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



Adaptive

Adaptive is only available for the modes **Percentage Reduction of Triangles** and **Target Number of Triangles**. With **Adaptive**, the coarsening algorithm minimizes the deviation of the coarsened surfaced 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.

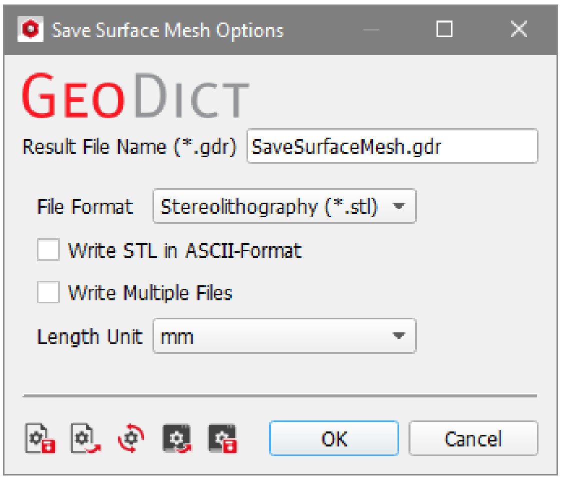
Save Mesh

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

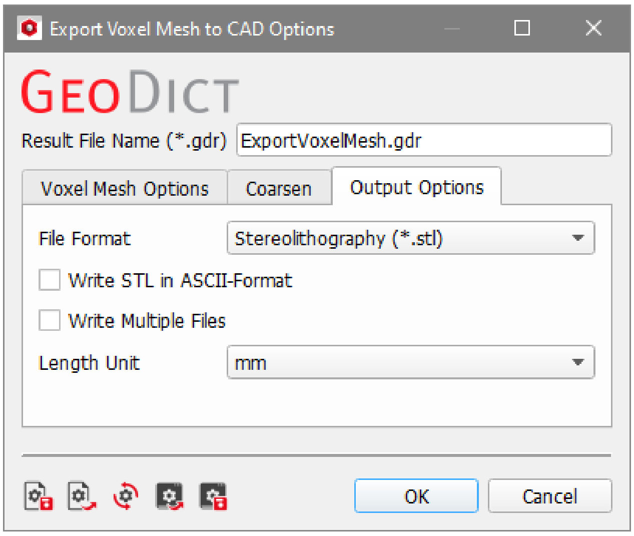
A dialog opens which allows to select the settings for the saved file. This dialog contains the same options as the **Output Options** tab in ExportGeo-CAD (see screenshots below).

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

Save Mesh options in MeshGeo



Output Options tab in ExportGeo-CAD



Result File Name

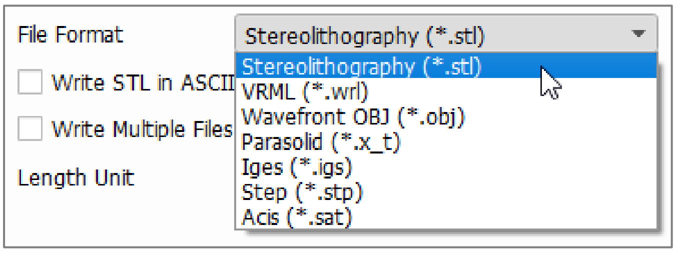
When saving a mesh, a result file is saved in the project folder and contains information about the selected parameters. Select the **Result File Name (*.gdr)** in the text box. The mesh files are also saved in the project folder.

File Format

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

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

If necessary, contact Math2Market for further information.

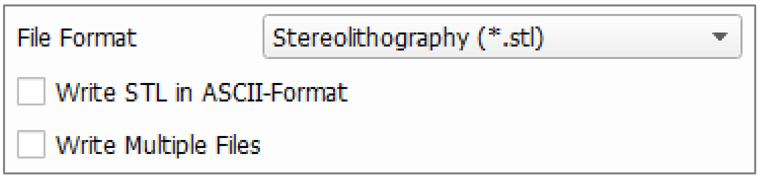


Additional options for the *.stl format

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

Write STL in ASCII-Format

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

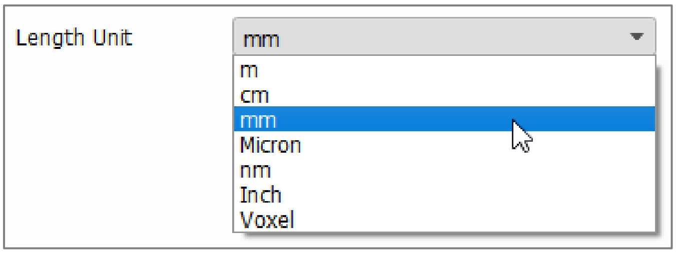


Write Multiple Files

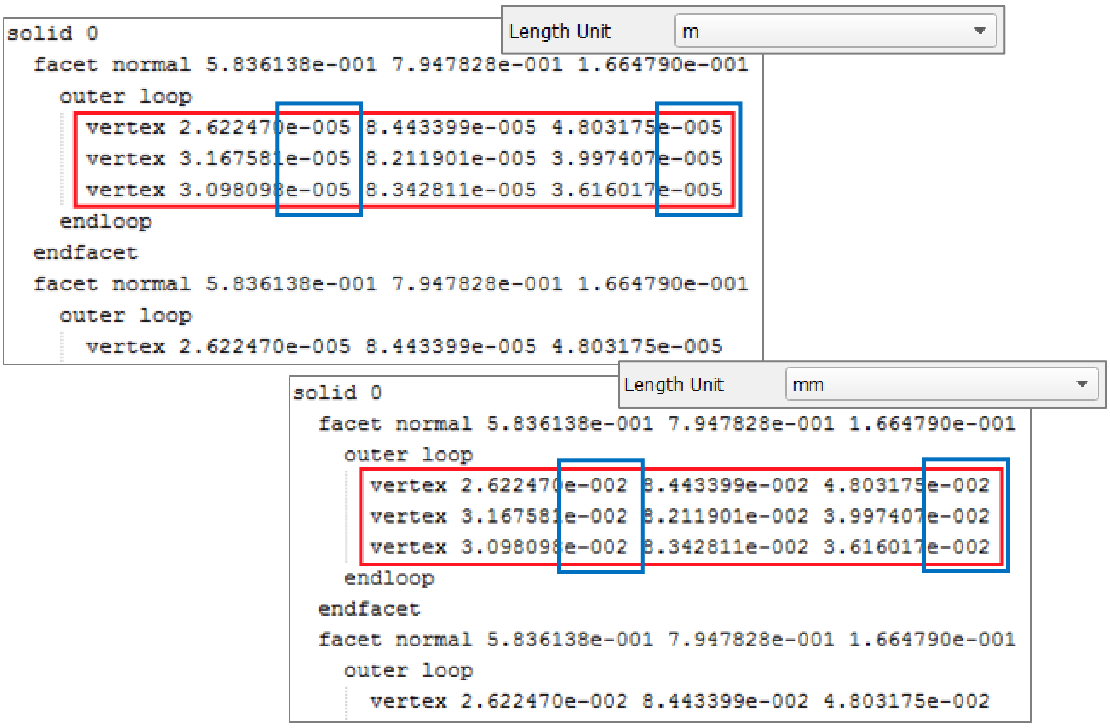
When the STL format is chosen and multiple meshes are currently loaded, these meshes can be saved to different files if necessary. This option is enabled with **Write Multiple Files**. For the other file formats, multiple meshes are always saved to one file.

Length Unit

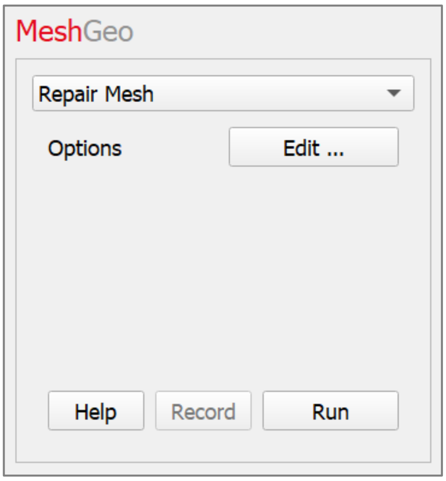
The unit in which the data is saved to the file can be selected with **Length Unit**. GeoDict can handle all these available units, but most external software cannot. Therefore, the unit should be chosen depending on the software with which the mesh is further processed.



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


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

Repair Mesh

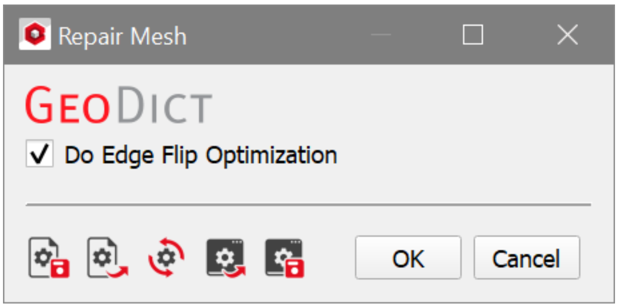


Repair Mesh provides options to repair meshes. The parameters for repairing the mesh can be defined by clicking the **Edit...** button.

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.

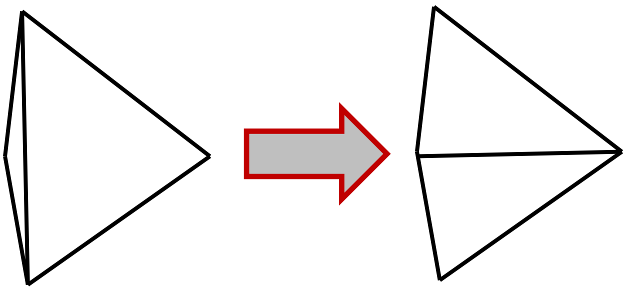
When all settings have been chosen, click **Run** at the bottom of the **MeshGeo** section to create the mesh.



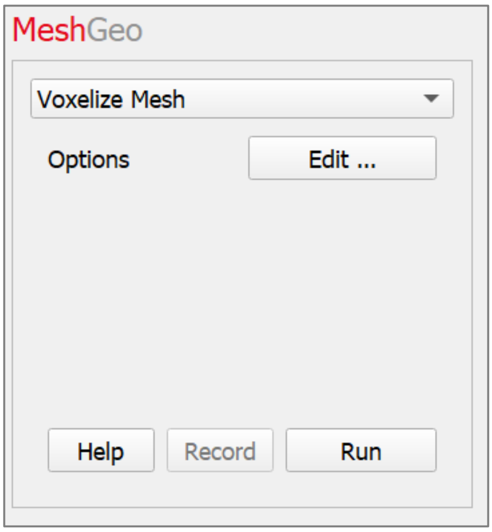
Currently, **Repair Mesh** contains only one setting (**Do Edge Flip Optimization**) which must be activated.

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.



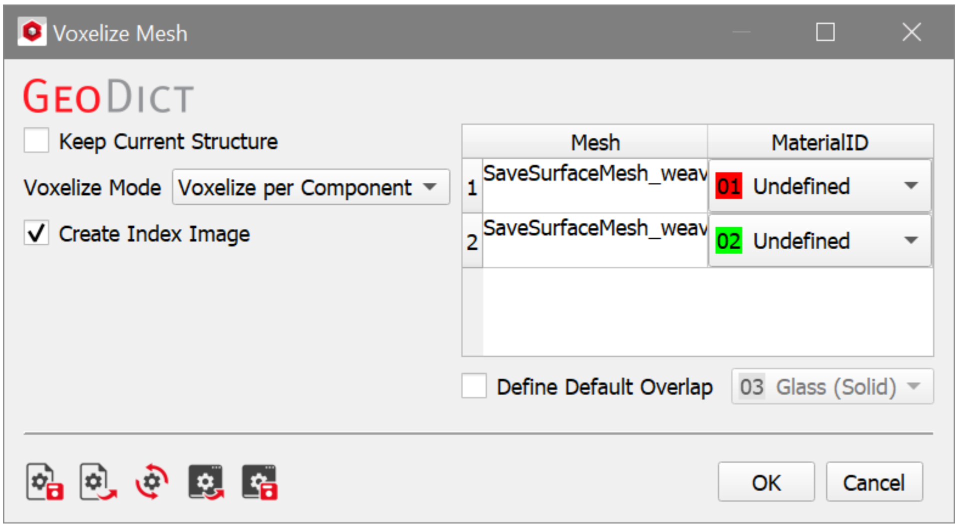
VOXELIZE MESH



With **Voxelize Mesh**, a voxel structure is generated based on the currently loaded meshes. The parameters for repairing the mesh can be defined by clicking the **Edit…** button. **Voxelize Mesh** provides the same functionality as ImportGeo-CAD, the only significant difference is that ImportGeo-CAD works on meshes during their import, while Voxelize Mesh works 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 **Run** at the bottom of the **MeshGeo** section to create the mesh.



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

https://doi.org/10.30423/userguide.geodict2022-exportgeocad

Technical documentation:

Sebastian Rief

Andreas Griesser

Barbara Planas

MATH 2 MARKET

Math2Market GmbH

Richard-Wagner-Str. 1, 67655 Kaiserslautern, Germany
www.geodict.com