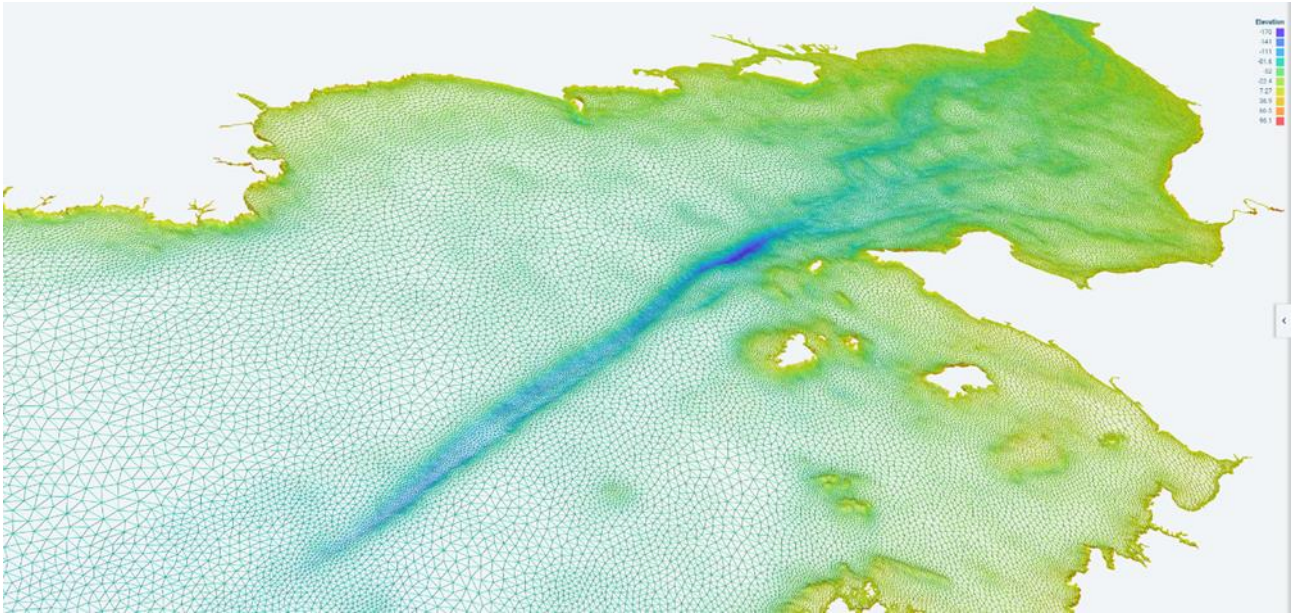


MIKE MESH BUILDER



Contents

1	Introduction.....	5
2	Getting started	6
3	MIKE Mesh Builder	7
3.1	About projects and workspaces	7
3.2	Creating a new Workspace	8
3.2.1	Supported file formats	10
3.2.2	Free-to-public data extraction.....	10
3.2.3	Projection systems	12
3.3	Using the Workspace	12
3.3.1	Layout.....	12
3.3.2	Editing a Workspace	14
3.4	Creating a geometry.....	15
3.4.1	The Geometries panel.....	16
3.4.2	Geometry selections.....	17
3.4.3	Duplication of geometries.....	18
3.4.4	Replacing a geometry	19
3.4.5	Applying operations to geometries	19
3.4.6	Editing geometries.....	22
3.5	Creating scatter data / Variables	23
3.5.1	Editing scatter data / Variables.....	24
3.5.2	Replacing a variable.....	24
3.6	Adding comments.....	25
3.7	Generating meshes	27
3.7.1	Configuring basic mesh parameters	27
3.7.2	Adding items to a Mesh Object	31
3.7.3	Removing items from a Mesh Object	32
3.7.4	Interpolating a mesh.....	32
3.8	Exclude items from mesh (holes).....	35
3.9	Analysing and improving meshes	36
3.9.1	Mesh post-operations.....	37
3.9.2	Mesh selections.....	39
3.9.3	Queries	40
3.10	Exporting data	41
4	Step-by-step mesh generation example.....	43
5	Problem solving.....	45
5.1	File won't upload	45
5.2	File doesn't work as expected	45
5.3	Maps won't load / visual inconsistency	45
A	Appendix: Generating meshes controlled by a background density function.....	46
A.1	Method description	46
A.2	Pseudo-algorithm	47
A.3	Examples.....	48

B	Appendix: Libraries used	49
C	Appendix: Non-supported projection systems.....	50

1 Introduction

The MIKE Mesh Builder (abbreviated MB in this guide) is a browser-based application for meshing that can be used online through Google Chrome and Microsoft Edge. It is currently not meant as a replacement for the MIKE desktop Mesh Generator but meant to explore the possibilities that the MIKE Cloud can bring to the meshing process.

MB is based in the DHI Microsoft Azure Cloud and supports meshing for scientific applications in various environmental flow domains. A mesh is produced from a combination of one or more polygons, polylines and/or points, called *geometries*. It can then be interpolated using *Variables*. MBs Cloud-based architecture allows multiple users simultaneous access for collaboration and parallel multi-tasking.

It produces a high-quality mesh file that meets the demands of simulation engines.

The MB philosophy is to combine decades of engineering knowledge with emerging modern technology, to enable ease of use, precision and speed.

This application is developed using the Agile process, meaning that updates to it are more frequent than they are for the desktop Mesh Generator. Be sure to communicate questions, remarks and wishes to DHI, as we are listening and improving our products based on your feedback.

2 Getting started

It only takes a few steps to get started with the MB application and obtain one or more meshes. The process supported by the MB application is as follows:

1. Login to the MIKE Cloud
2. Select a project and add any files you may want to use, then select the MB app (see 3.1)
3. Select an existing Workspace or create a new Workspace (see 3.2 and 3.3)
4. Create geometries or use them when you have imported them (see 3.4)
5. Modify your geometries, using available vector operations (see 3.4.5)
6. Create variables if you don't have any available from files (see 3.5)
7. Create one or more meshes by selecting data and control parameters, then interpolating them (see 3.7)
8. Analyse meshes using quality measures (see 3.7.5)
9. Apply post-meshing operations, such as refining or smoothing (see 3.8.1)
10. Export meshes to Data Admin (see 3.8.2) and download them for further use.

3 MIKE Mesh Builder

There is one general page for the MIKE Cloud where all the applications that you have access to can be found: <https://home.mike-cloud.com/>

We recommend using this page as a bookmark, as this page also holds the announcements regarding the applications. That means you can find information regarding downtime and warnings about these applications here.

Once you create a project, you can then select the application Mesh Builder:

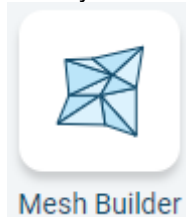


Figure 3.1 The MB application button on the Cloud Homepage.

After you click the button, the application should be started and you should see the *Workspaces* page for the selected project.

To switch to another project, it's possible to click 'My projects' at the top of the screen:



Figure 3.2 Breadcrumbs (on the right) in the top bar

This will take you to the *Projects and recent workspaces* page. Clicking a project from there will take you to the *Workspaces* page for that project.

3.1 About projects and workspaces

After logging in, you enter the *Projects and recent workspaces* page. This page is relatively empty when you enter it for the first time but will fill up with use.

But what are projects, workspaces and how are they connected?

The MIKE Cloud landscape enables applications to work together. The MB uses the functionality offered by the MIKE Data Admin application, for storage and conversion of files that are uploaded. The data ends up in a **project**.

The data in a project might or might not all be relevant for a mesh you want to create. It may or may not need to be altered for it to be useful for your purpose. For MB use, this uploaded data can be grouped together differently, using **workspaces**.

The *Projects and recent workspaces* page (Figure 3.3) enables you to work with these. The page is divided into three parts:

- At the top of the screen is a Search bar. It enables you to search in the Projects that exist and are available to you through the Data Admin application.
- Below that are recent workspaces you have interacted with (can be empty at first login).
- At the bottom is a list of projects that are available to you through the Data Admin application.

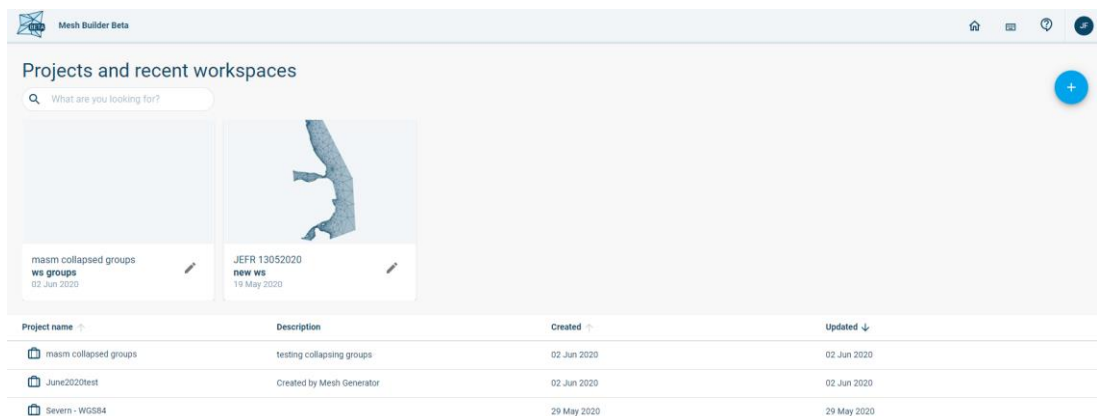


Figure 3.3 The Projects and recent workspaces page

If you are looking for an existing project, you can do so from the Search bar or by looking through the list at the bottom. If you click on a project, you will see the available workspaces in that project. You can also create a workspace from there, using the button on the top right.

If you want to create a new workspace, hover over the button at the top right. It enables you to create a new workspace in an existing project or to create a new project in the Data Admin application and add a workspace to it.

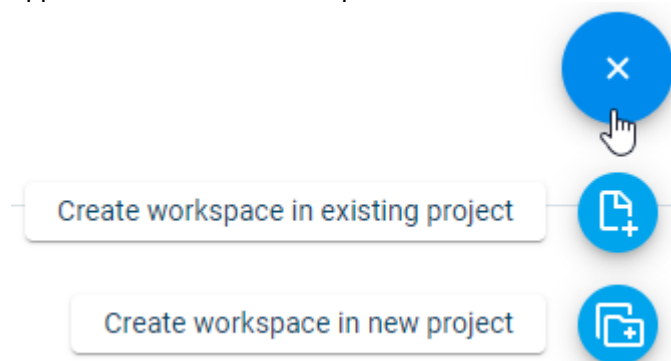


Figure 3.4 The + button enables you to create a new workspace or both.

Note: Be aware that at this moment, it is not possible to upload data directly from the MB, so if you want to do this, it is probably easier to create the project entirely from the Data Admin application. The option *Create workspace in new project* should only be used if you want to start by using extractions for data (see 3.2.1).

3.2 Creating a new Workspace

Once you have clicked the button to create a new workspace (see 3.1), a new panel will open on the right side of the screen (see Figure 3.6):

1. Project selection
Your selected project is preselected here. If you choose to change it, the workspace will be created in the project you change it to.
2. Workspace name and description (optional)
You are free to name your workspaces as desired, and an optional description can be added. If name is left open, the name will be *Untitled*.

3. Data from project

Tick the checkboxes for the data you want to use for the mesh you are about to create. Will only appear if the selected projects holds relevant data in the Data Admin application. Also, if no ticks are set, no data is put into the workspace (this can be changed later).

Note: see 3.2.1 and 3.2.3 for information on restrictions regarding imported data.

4. Data type declarations:

Some dataset types need user definition of their content:

- a. The *geometry* type should be recognized automatically, provided the source data is a standard shapefile. Items with this type will be used as Geometries to set up the basic shape of the mesh.
- b. The *variable* type will not be automatically recognized. Items marked as such can be used as for interpolation.
- c. The *mesh* type should automatically be recognized, provided it is in Kitware's VTU format. These items will enter the MB as previously generated meshes.

5. Projection system

The dark EPSG code above the search option is the projection system that will be used to create the workspace. The default is **EPSG:4326 WGS 84**.

The dropdown/search bar underneath it will determine what the projection will be for your workspace. It allows typing of name and EPSG codes and will show the results. Below the dropdown recently used projection systems will be shown, based on recently used workspaces; clicking any will change the projection system for the Workspace to be created to the selected option.



Figure 3.5 Projection selector and recently used projection underneath

If the projection system is different from the one(s) in the selected *Data from project* (point 3 above), an attempt will be made to convert the selected files to the projection system picked here; however this is not guaranteed to work under all circumstances.

Note: It is currently not possible to change the projection system after the workspace has been created.

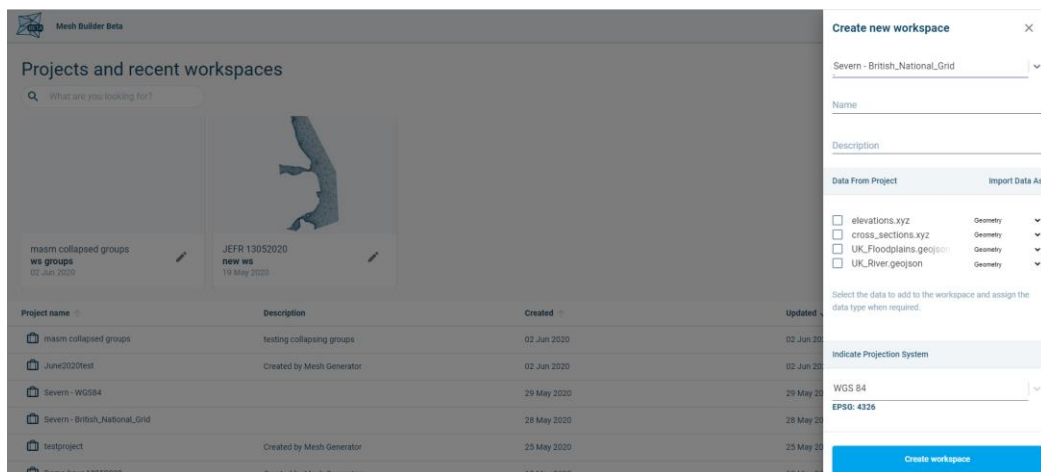


Figure 3.6 The Workspace creation window and its related user-actions.

Click the *Create workspace* button once you are done.

Upon Workspace creation success, you enter the Workspace page.

3.2.1 Supported file formats

Not all file formats are (fully) supported yet. The current version only supports the following data formats:

- Geometries: ESRI shapefiles (with standard association: *.shp),
- MIKE internal formats: *.dfs0, *.dfs1, *.dfs2, *.mdf,
- Raster variables: ESRI ASCII Grid format (*.asc),
- Meshes: Kitware VTU (*.vtu), DHI MIKE Zero (*.mesh),
- Other: *.tif, *.xyz, *.vti

This list will grow in time.

3.2.2 Free-to-public data extraction

The Mesh Builder supports access to free-to-public data providers. The amount of data available differs per region and is not subject to screening by DHI.

A project and workspace are required for the extraction. Once those are created and no other data is present, MB will offer the option to extract data:

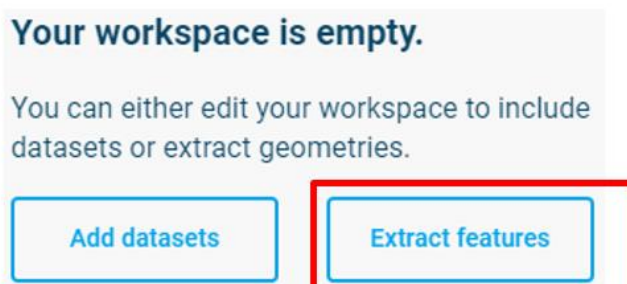




Figure 3.7 Add datasets or extract features in an empty Workspace

If you want to use the data extraction in a project that already has some data in it, click the  tool on the left menu while no other items are selected on the right.

Zoom to the area of the map that you wish to extract; or – if you have a polygon representing the area of interest – select that from the geometries on the right (this allows you to skip the next two steps, continue under the note, below).

- Click with the active tool to select your first point, then proceed to draw lines and click until you have the area you want to extract.
- Double click to close the selected area. A closed polygon should appear.

Note: for more options such as deleting a point, look under the keyboard shortcuts () in the top right.

Once the area is selected, select a data provider that you would like to extract from the panel on the right.

The following data providers are available:

- **OpenStreetMap:** The OpenStreetMap provider enables retrieval of GIS vector representations of land surface features, including elements of the natural and built environments. More information about the data can be found [here](#). The following data sets are available for extraction:
 - Coastline
 - Road
 - Building
 - River bed
 - River path
 - Lake
 - Landcover
- **NOAA:** The NOAA data provider enables retrieval of GIS vector representations of coastlines. High-, medium-, and low-resolution data sets are available. More information about the data can be found at <https://shoreline.noaa.gov/>.
- **GEBCO:** The GEBCO data provider enables retrieval of raster bathymetry data. The 2023, 2022, 2021, and 2020 data sets are available. The 2023 data set includes a 'sub-ice' option that uses land surface elevations for areas covered by permanent ice. More information about the data can be found at https://www.gebco.net/data_and_products/gridded_bathymetry_data/.

Once the area and type are correct, click Extract. At this point, changes can be made before a geometry is created, so Simplify and Dissolve operations can be used on the extracted data, and drawing tools on the left can be used to change and add points to the extraction.

When you are satisfied with the extracted item to be added to the Geometries, click Create.

The extraction may take a while, depending on the area and the amount of data, and will then be added as a geometry.

This extraction can be done multiple times in a single area to get multiple types of data.

3.2.3 Projection systems

The Cloud platform currently supports over 9000 different projection systems. There are currently 28 projection systems that are currently not supported (see Appendix: Non-supported projection systems). Please specify the projection system of the file you are uploading in Data Admin, as it allows MB to correctly use this information.

The projection (EPSG) code of the workspace is reported in the bottom bar of the screen once you enter the Workspace (see Figure 3.8). For information on how to set the projection system at workspace creation, see Section 3.2.

3.3 Using the Workspace

The workspace in MB holds most of the functionality. This guide will first describe the layout of the page, then go in depth about the functionality.

3.3.1 Layout

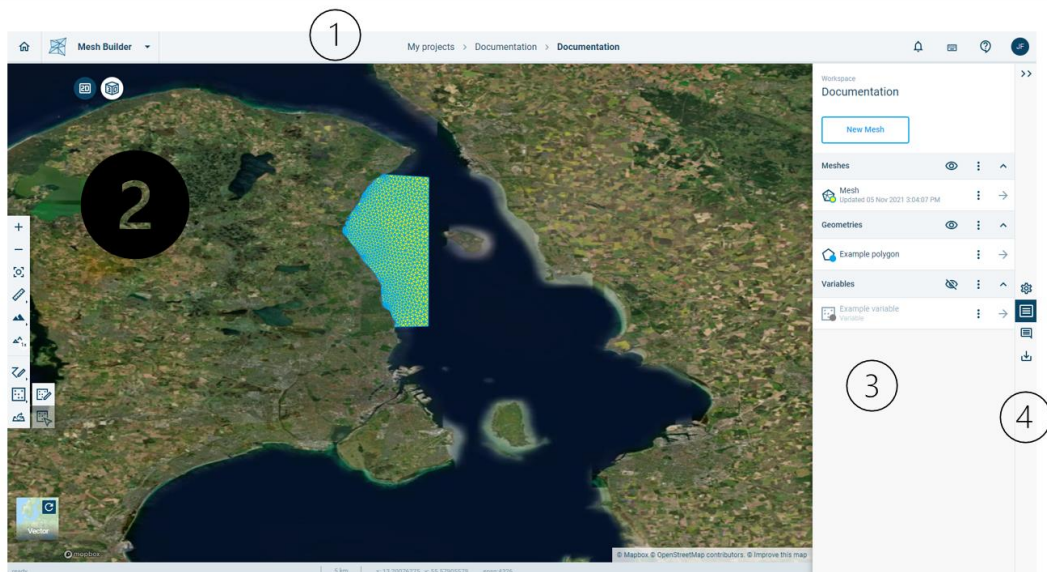


Figure 3.8 The Workspace page with its basic elements and with default plan view of the available geometries.




Figure 3.8 shows the 4 main elements of the Workspace page:

1. Top bar
2. Map area
3. Data panel
4. Navigation toolbar

Top bar












- On the left is a quick switch to the other Cloud applications you have access to. If you select an application from the list, it will open the application in the project you are currently working in in Mesh Builder.
- In the middle of that is the navigation to get back to the *Projects and recent workspaces* page (My projects) and the workspaces in the current project (displays the name of the

current project). To the right of that are five buttons:

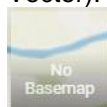
-  notifications will be shown here
-  view the keyboard shortcuts that can be used in the map area
-  find the user guide, example data or send an e-mail to support
- A button to the right with your initials. Click it to find a button to Log out

Map area

The second, below the top bar, on the left, is the map area. There are several buttons here, as well as a toolbar. The toolbar will change, based on what you have selected, to support actions that you can take.

-  switch between 2D and 3D view. The active one is the one in dark blue.
-  Zoom in
-  Zoom out
-  Reset view
-  Measurement tool (choose to measure between two points or an area)
-  Choose 3D terrain
-  Choose the zoom ratio (2x, 5x, 10x, 25x, 50x)
-  Draw geometry (Draw rectangle, Draw circle, Draw polygon, Draw polyline, Draw point)
-  Draw Variable (Draw variable, Edit variable)
-  Extract features from open data sources
-  View info on extracted Landuse tool (only available after extracting Landuse or when editing a shape).

It is also possible to change the map used in the background. This can be done by clicking the button on the bottom right. It indicates the option you will get it (No Basemap, Mapbox Sattelite, Vector):









Data panel

The third major component, to the right of the map area, is a panel with all data saved in the workspace, including Meshes, Geometries and Variables.

Above the data panel, there is a display of the workspace name along with a context menu just above the name. The context menu provides access to three options:






1. Operations: This option provides access to different options for creating a polygon that can be used to define an outer boundary (domain) for meshing.
2. Send snapshot to support: This will create a “snapshot” of a workspace and save it in folder that is accessible to the DHI support team. A snapshot is a version of workspace that can be saved and restored later. Sending a snapshot to support enables the support team to reproduce issues reported in support cases. If a bug is found, the snapshot also enables the team to debug and test solutions.
3. Create workspace snapshot: This will create a “snapshot” of a workspace and save it in the same folder as the workspace. A snapshot is a version of workspace that can be saved and restored later. This option is useful for saving one’s work if, for example, one would like to pause a subscription to Mesh Builder.

Below the workspace title, data are organized under the headings Meshes, Geometries, and Variables. If one clicks on the title of a data set, you enter the properties of the selected data set, which are then displayed in the panel. Otherwise, the following icons are visible:

-  Hide all items of this type on the map (hidden items will not be loaded upon loading of the workspace, only when interacted with such as clicking or using them in an operation)
-  Show all items of this type on the map
-  Open the context menu for an item (or a group of items when it is in the header). Properties can also be accessed through the context menu.
-  Collapse this section
-  Unfold this section
-  Go inside this item


Navigation toolbar

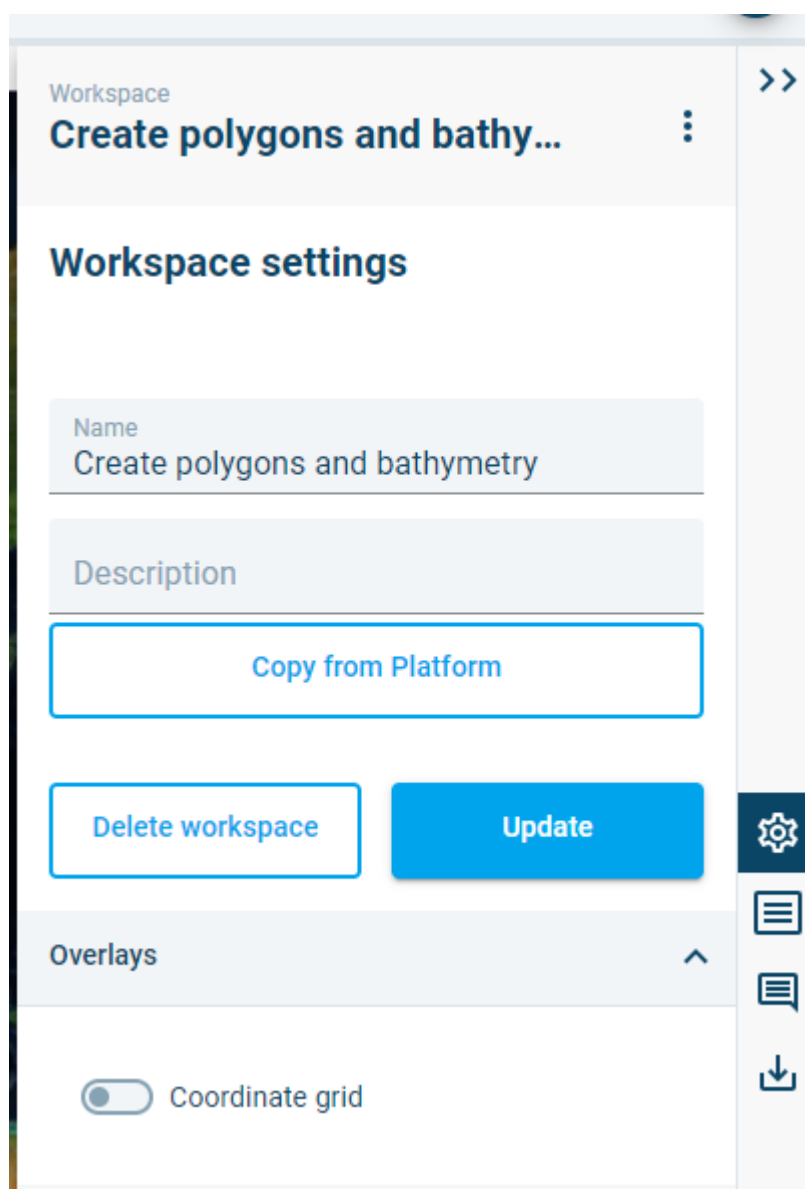
The fourth and last component, on the uttermost right side, is a navigation toolbar:

-  Collapse the panel. Clicking any of the other buttons in this toolbar unhides the panel menu if it is hidden.
-  Edit Workspace settings
-  Go to the overview of Meshes, Geometries and Variables
-  Go to the Comments panel
-  Go to the Export panel

3.3.2 Editing a Workspace

The Edit Workspace settings panel is used to copy geometries and variables from Cloud Admint into the Mesh Builder. It is also possible to change the Workspace name and its description.

Click the “*Edit Workspace settings*” button () in the right navigation panel.



The Name and Description fields can be used to edit the workspace name and, optionally, add a workspace description.

The Copy from Platform button is used to select data from Cloud Admin to copy to the Mesh Builder workspace. Datasets are copied from Cloud Admin; in other words, changes to data in the Mesh Builder do not affect the original data saved in Cloud Admin. After selecting data to copy, the user is prompted to specify whether each data set to be copied is a mesh, geometry, or variable. More information about meshes, geometries, and variables is given below.

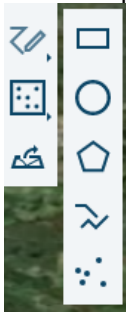
A delete button is available for deleting a workspace. It's also possible to turn on the coordinate grid for the workspace projection.

3.4 Creating a geometry


If you want to draw your own geometry instead of uploading a file or extracting data, go to the Workspace and click on the tool in the left bar:



This will open five options:





- Draw rectangle
- Draw circle
- Draw polygon
- Draw polyline
- Draw point


Once you click one of them, your mouse pointer will turn into a drawing tool that can be used on the map area. On the right, the panel will display a summary of what you have done. Clicking the Cancel button will cancel the tool and you will be able to use your mouse for navigation again. It is still possible to click the  button on the top right to view the keyboard shortcuts.

Note: Circles are drawn from their center.

Note: A polygon or polyline needs to be completed (double click or press space bar at the end of them) for them to appear in the panel to the right.


Note: It is possible to freehand a polygon or polyline by pressing the CTRL key while moving the mouse.


It is possible to rename a geometry after creating it, by clicking the menu item  behind its name from the main Workspace page and clicking  **Rename**.

Click on the name and change it. Click  once you are done. The item will be renamed.

We will now go into more detail for this panel, called the Geometries panel.

3.4.1 The Geometries panel

The geometries panel is displayed in Figure 3.9 and can be found on the right side of the screen after clicking the  button for any item under geometry.

The panel has an action menu to the top right () and further consists of multiple tabs (in the example Figure 3.9 this would be Overview and Selections), the option to exclude it from the mesh (considering it a hole), an area called Geometry Analysis and an area called Visualisation Settings.

The action menu gives all the available options that can be used for this Geometry.

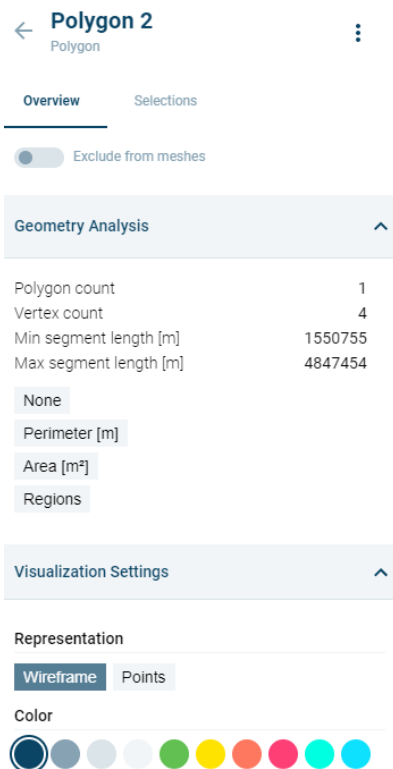


Figure 3.9 Geometries panel: Plotting of metadata controls and accessible actions.


The Geometry Analysis area enables you to look at certain data more in depth. It shows polygon and vertex count and segment lengths, but also other variables that can be clicked and reviewed (for more info, see 3.7.5).

The Visualisation Settings make it possible to change the appearance of the Geometry in the map area: when clicked, it can change from wireframe to points and can change the colour.

3.4.2 Geometry selections

The Selections tab in the Geometries panel shows spatial selections made for the selected geometry and provides functionality for creating new selections. When creating a mesh using a geometry, selections enable assigning different properties to the selected area, such as finer or coarser meshing. Selections can also be used to specify open boundaries.

Spatial selections can be initiated by clicking on the button, 'Make a spatial selection' or by

clicking on the Spatial selection icon () from the left menu. Once the selection is initiated, a selection can be made by drawing by hand or by selecting another geometry. A selection can be made inside a polygon geometry without intersecting any line segments. of the geometry.options while in this panel.

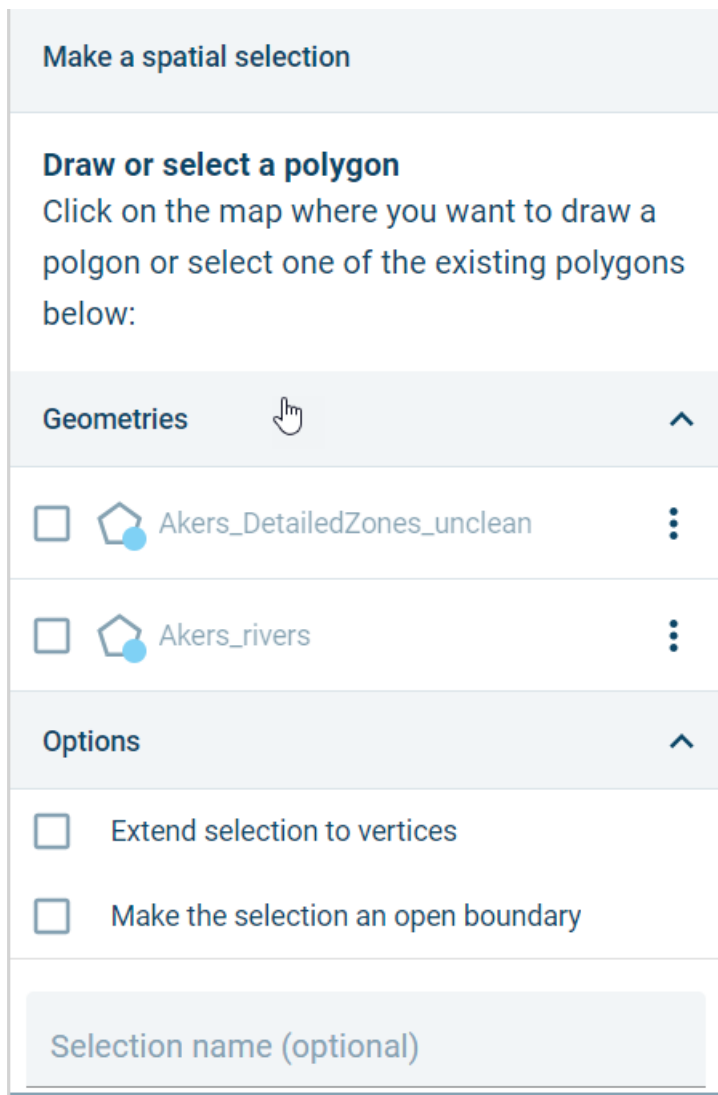



Figure 3.10 Making a spatial selection

The options, 'Extend selection to vertices' and 'Make the selection an open boundary' can be used to create open boundaries when using a geometry in a meshing operation. 'Extend selection to vertices' will extend the portion of any line segment included in a selection to the next vertex outside the selection area. 'Make the selection an open boundary' will, during meshing, assign open boundary node numbering to mesh nodes located on segments within the selection area.

3.4.3 Duplication of geometries

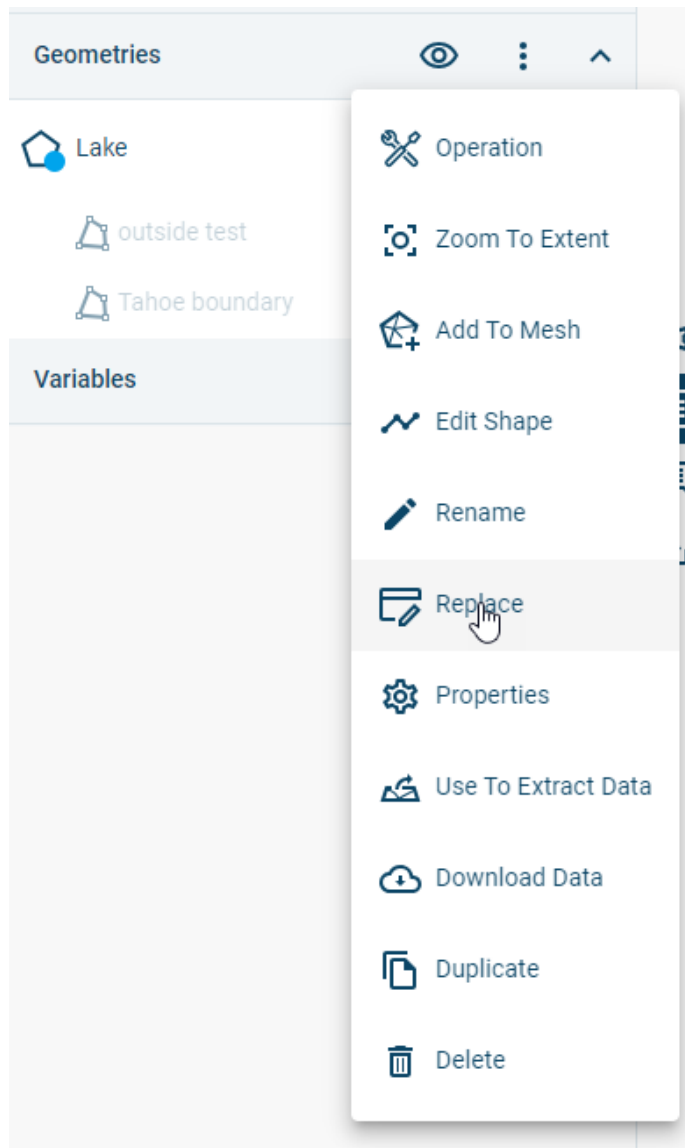
Geometries and meshes can be duplicated. From the main Workspace page, go to the item to be duplicated and click the menu item  behind its name.

Click  Duplicate .

The duplicated item is added.





3.4.4 Replacing a geometry

A geometry can be replaced with another geometry from Cloud Admin without changing the name or other properties of the geometry. In this way, geometries can be 'updated' from outside the Mesh Builder. To replace a geometry, click on the three-dot icon to the right of the geometry name and select "Replace".



3.4.5 Applying operations to geometries

Operations change the structure of geometries. Operations can be accessed as follows:

1. from the Workspace page on the right panel at the geometry level, by pressing  and then picking  Operation. This will lead to a selection menu where the geometries that the operation should be applied to can be selected.
2. from inside the geometry, use the  menu on the top right, then choose  Operation.

For some geometry operations, it's possible to limit the operation to a subset of the selected geometry using the Select Attribute field. The following options may be available, depending on the selected operation:

- All: Entire geometry
- Id: Selected geometry IDs. Only a range can be selected.
- Area: User-specified range of polygon areas.
- Perimeter: User-specified range of polygon perimeters.
- Elevation: User-specified range of elevation attribute values
- Selections: If a selection has been defined on the geometry, it's possible to limit the operation to the selected areas.
- Geometries: It's also possible to use another geometry to define the spatial extent of the geometry operation.

If a geometry has other attributes, it's also possible to limit an operation to range of values for one of the other attributes.

By default, an operation will create a new geometry, preserving the original. However, it's possible to overwrite the original by disabling the switch, "Create geometry copy".

The following operations are available:

Aggregate

This operation will combine existing polygons based on the indicated distance between them and how different the new shape is allowed to be.

The operation has three parameters that can be used to define the outcome:

- *Distance between original polygons* in meters indicates how closely together polygons need to be in order to be aggregated. Default is 100m
- *Polygon outline fidelity* in meters determines how closely the aggregated polygon will follow the outlines of the original polygons. Default is 40m.
- *Percentage change of polygon outline* in % determines how much the outline of a polygon is allowed to change in the operation. Default is 0%.

Buffer

This operation creates a new polygon that is bigger or smaller than the selected polygon. A value in meters needs to be entered for the operation to work. A negative entry (i.e. -10) will cause the polygon to shrink. Default is 100m.

Clip

This operation enables you to create new geometries from existing polygons by selecting the 'subject' you want to have a part of and then defining a 'clipper' polygon by which the subject is then clipped. The selected operation determines the behaviour:

- Intersection: clips the selected polygon where it is intersected by the clipper polygon. A new geometry is created from the area where clipper and subject intersecting areas.
- Difference: clips the selected polygon where it does not overlap the clipper polygon. A new geometry is created from the areas where there is no overlap with the subject.

Densify

This operation adds more vertices to the geometry by splitting its segments. A value in meters needs to be entered that determines where the split will occur. Example: 100m will split segments every 100m of their current length, whereas 30m will split them every 30m of the length, creating more vertices. Default is 100m.

By default, this operation will keep the original geometry points. It is possible to change this by clicking the switch in front of *Keep geometry points* (the dot should be on the left to turn this off).



Extract by attribute

This operation creates a new geometry as a subset of the geometry that the operation is performed on. Select an attribute (i.e. Region, ID or Length) and define the minimum and maximum values to specify what to include in the new geometry.

Polygonize

This operation will try to turn multiple polylines into a single polygon (only available for geometries of type Polyline).

Quick Simplify

Reduces the number of vertices of the geometry by applying a Douglas-Peucker algorithm. A slider is used to make it easy to see the impact of different values on the simplified geometry.

Simplify

This operation reduces the number of vertices of the geometry. Two options are available.

- A Douglas-Peucker algorithm, which requires setting a tolerance in meters.
- An option that removes all segments smaller than a user-specified distance.

Simplify buildings

This operation reduces the number of vertices of the geometry and is targeted at simplifying shapefiles depicting building footprints in an urban setting. The logic is based on an algorithm proposed by Gwanyong Park, Changmin Kim, Minhyung Lee and Changho Choi in the following paper:

<https://www.mdpi.com/2076-3417/10/16/5425>

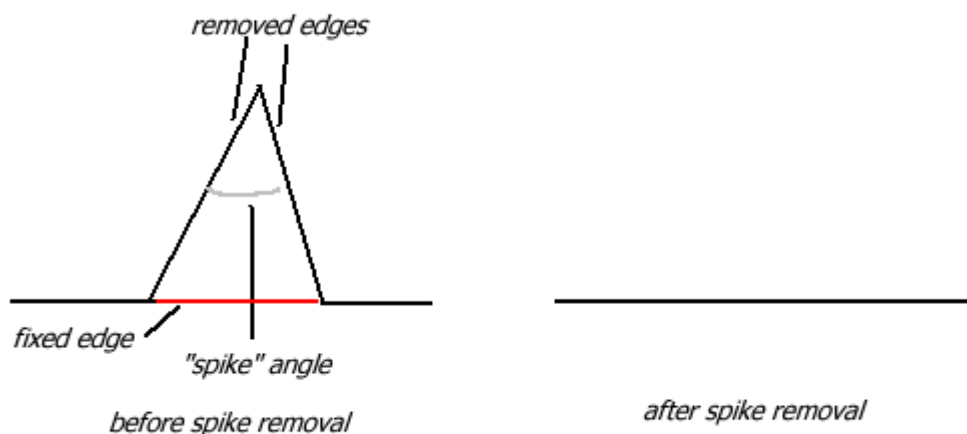
The algorithm removes “insignificant” edges by checking the perpendicular distance between the centres of each edge pair and angle between edge normals. The following parameters are used:

- Simplification distance (meters): Minimum perpendicular distance between edges. Default is 5 meters.
- Simplification angle (degrees): Minimum angle between edge normals. Default is 30 degrees.

If the distance or angle between edge pairs is less than the simplification distance or simplification angle, then the shorter of each pair is removed and a new building perimeter is constructed from the remaining edges.

The following additional parameters are available.

- Snap distance (meters): The operation will attempt to connect buildings separated by less than this distance. Default is 0.5 meters.
- Spike angle (meter): The operation will attempt to remove ‘spikes’ in building edges with an angle less than this value. Default is 30 degrees.



- Remove holes: If enabled, this switch will remove polygons that are entirely enclosed by other polygons. The switch can be used to remove courtyards and other enclosed areas. Default is on.

Smooth

This operation smooths the geometry path. A value must be entered for the number of control points to be added to each input vertex to create a smooth curve. Default is 5.

If the option of keeping the geometry points intact is chosen, additional control points are interpolated by the Catmull-Rom algorithm. If not, then the points are approximated by Chaikin's algorithm.

Split

This operation splits an existing geometry into two or more geometries. A splitting polyline must also be specified.

Union


This operation enables the creation of one new polygon from several existing polygons.

Split by attribute


This operation enables the splitting of a geometry by one of its properties. Currently the limit of geometries generated from this is set at 25, so if the operation would generate more geometries it will fail.

3.4.6 Editing geometries

Geometries can be edited, either by moving existing vertices; adding vertices; adding new polygons, lines, or points to existing geometries; or with editing operations.

Geometry editing is initiated by clicking on  next to the Geometry name in the right-hand panel and then selecting, "Edit Shape". The selected shape can then be edited in the following ways:


- **Add vertices:** Vertices can be added to polygons and polylines by clicking anywhere between two vertices. The newly created vertex will be displayed on the map, along with the other vertices.

- **Move existing vertices:** Existing vertices can be moved by clicking and dragging with the mouse button held down.
- **Adding new polygons, lines or points:** New polygons, lines, and points can be added to a geometry by clicking anywhere besides the existing geometry. By default, the same geometry type (i.e., polygon, line, or point) will be created. However, it's possible to create a mixed geometry by selecting one of the other types from the geometry creation tool  in the left-hand menu.
- **Editing operations:** The clip, simplify, and dissolve operations are also available in the editing menu for convenience. See section 3.4.5 for a description of these operations.

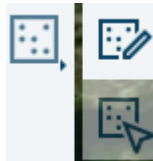
3.5 Creating scatter data / Variables

The data category, “Variables”, refers to scatter data and raster data.

Raster data can only be copied from Cloud Admin, and can't be edited after copying to the Mesh Builder.

Scatter data can be copied from Cloud Admin or created in the Mesh Builder. To create a scatter data variable, click on the tool in the left bar: 

This will open two options; the bottom one will only be active if a variable is present and selected when the tool is selected:



- Draw Variable
- Edit Variable

Once you click on the tool, your mouse pointer will turn into a drawing tool that can be used on the map area. On the right, the panel will display the value that the drawn point will get. Simply change it to the desired value and click the map to create a point with that value, and change it again and click the map to make points with different values.


Draw variable

Elevation value

0

Figure 3.11 Elevation value for newly drawn points will be 0 with this setting

Clicking a drawn point and pressing the *Delete* button on the keyboard will remove a drawn point. Clicking a drawn point and then holding and dragging the mouse button while pressing the *Shift* button on the keyboard at the same time allows you to move a drawn point.

When you have drawn all required points, click the  button in the panel, and all points will be grouped together into a Variable, shown on the right of the screen. The variable will initially not be shown after adding it.

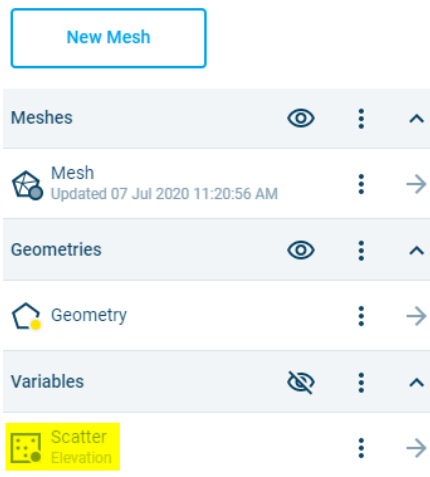







Figure 3.12 Newly added Variable drawn with the Draw Variable tool

Clicking the Cancel button will cancel the tool and you will be able to use your mouse for navigation again. No variables will be added.

It is possible to rename a variable after creating it, by clicking the menu item  behind its name from the main Workspace page and clicking  **Rename**.

Click on the name and change it. Click  once you are done. The item will be renamed.

3.5.1 Editing scatter data / Variables

It is possible to edit scatter data in existing variables by selecting the menu item  behind its name and then selecting  **Edit Data**.

This will open up the Draw Variable tool, allowing you to add more points to the existing variable. Clicking a drawn point and pressing the *Delete* button on the keyboard will remove a drawn point from that variable. Clicking a drawn point and then holding and dragging the mouse button while pressing the *Shift* button on the keyboard at the same time allows you to move a drawn point in that variable.

In order to change a value on a point, select the Edit variable tool on the left menu instead



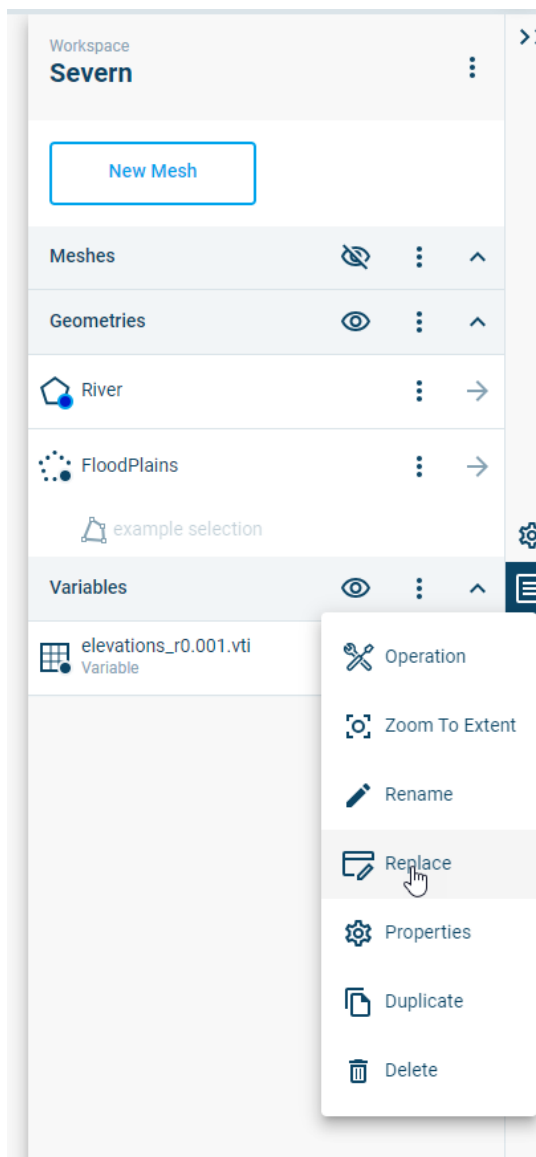
This tool will allow you to select a point and change the value from the right panel. Once you are satisfied with the new value, click:



The value of the point will be changed.


3.5.2 Replacing a variable

A variable can be replaced with another variable from Cloud Admin without changing the name or other properties of the variable. In this way, variables can be 'updated' from outside the Mesh Builder. To replace a variable, click on the three-dot icon to the right of the variable name and select "Replace".



3.6 Adding comments

Comments can be used to communicate to colleagues working on the same project, to point out improvements or as reminders to yourself.

The comments panel can be found on the right side of the screen after clicking the . It initially gives an overview of the comments that are currently in this Workspace (may be empty if no other comments exist).

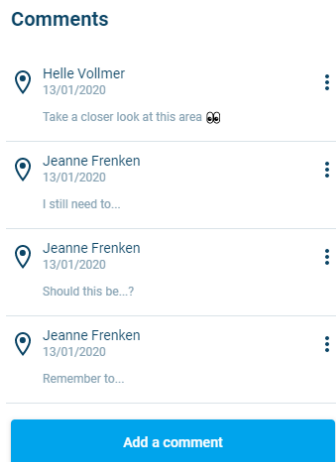
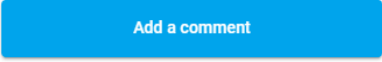




Figure 3.13 Comments panel

To add a comment, click the button  at the bottom or click on the  tool in the left toolbar and then click on a point on the map.

A dot is projected where the comment marker will appear and the panel to the right changes so you can enter the desired text. If the placement is wrong, simply click again to place the comment marker elsewhere.

Click on the Comment line to the right to type in your comment.

It is possible to use the standard Windows emotes here, by pressing Windows key ( on your keyboard) and the dot at the same time. A pop-up will appear, where you can add the desired emoticon by clicking on it.

At the bottom of this pop-up are categories with emoticons, and you can type a search term at the top.

Click anywhere next to the pop-up to close it again.

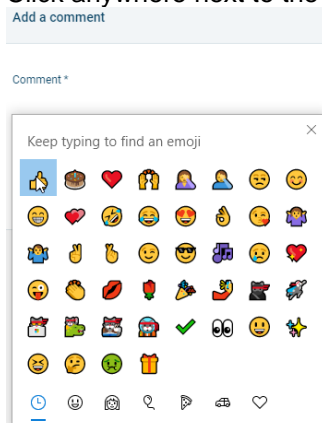





Figure 3.14 Emoji pop-up

Once the comment is as desired, click . A comment marker is added, and the comment is added to the list of comments. All users able to see the Workspace will now be able to see the comment.

In order to delete a comment, click the menu behind it () and then click  Delete. The comment will be deleted.

3.7 Generating meshes

Once the geometries and variables are set up, it is possible to create a mesh.

From the main workspace page, click the button on the top of the panel *New Mesh*:



The first step is to determine which geometries should be added to the mesh, as not all geometries in the workspace might be relevant. The right panel shows an overview of the available geometries in the current workspace.

Tick the boxes of the geometry items that you want to add:

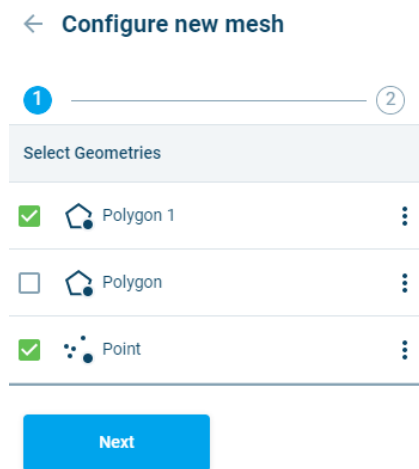



Figure 3.15 Selecting input items for further configuration and mesh object creation.

As a visual aid, the selected items will turn a different colour in the map area.



Click  once all the geometries are selected.

3.7.1 Configuring basic mesh parameters


The panel switches to the *Configure new mesh* panel. Within this panel, it is possible to make changes to some general mesh settings, as well as settings for each geometry.

All algorithms in the MB application performing *constrained Delaunay quality triangulations* are based on minimization and maximization principles regarding triangles shape and size. The algorithms attempt to maximize the minimum angle between two edges of a triangle (α_{min}), while making sure that the smallest angle in the triangulation is least as large as the smallest angle in any other mesh triangle. It also minimizes the triangles edge-length (d_{max}).

The general mesh settings are:

- Min angle; this is the minimum dihedral angle - α_{min} , a limit on the lower bound of the angle formed by two adjacent edges of a triangle in the generated mesh. The default value is 26 degrees.
Note: This control parameter has the highest effect on mesh quality. Values bigger than 30 degrees are known to likely generate mesh failure and should generally not be used.

- **Snap tolerance:** this parameter specifies a distance in meters below which vertices of the geometries to be meshed are snapped to the same location

In order to configure a geometry, click the  button next to it.

Five meshing options are available: Triangulate (default value), Rectangular (mostly used for offshore), Channel (mostly used for channels), Radial (for meshing a radius around a point), or Coastline (used for creating triangular meshes adjacent to coastlines). Additional specifications are made depending on the choice of mesh operation.

Triangulate

The triangular mesher is available for all geometry types.

- **Size criteria:** Users have a choice of two criteria for specifying the minimum size of mesh elements: area or edge length.
- **Maximum element face length (m) or area (m²):** This limit is applied throughout the meshing area.
- **Element length (m) or maximum area (m²) at mesh border:** This constraint is applied along polygon borders and/or polylines.
- **Use only vertices:** If this switch is turned on, only geometry vertices will be inserted into the mesh, while line segments will be ignored.
- **Enable gradual size:** If a different sizing criterion is applied at the mesh border, this switch will enable a gradual transition.
- **Use background density function:** This switch enables use of a variable to control the density of mesh elements. More information is available in Appendix A. This switch activates the following input parameters:
 - **Number of points (default 10.000):** This is the number of mesh nodes to be generated by the density function.
 - **Buffer size:** This parameter excludes a buffer adjacent to the geometry from the densification operation.
 - **Select a variable or no variable:** This field enables selection of variable to control the density of mesh elements. The density of elements will be increased in areas with larger values. It's also possible to use the density function without a variable. If a variable is selected, two additional switches can be applied:
 - **Apply gradient:** This switch uses a gradient instead of actual values to drive densification, with higher density applied in areas with larger gradients.
 - **Invert the density function:** This option will increase the mesh density in areas where variable values are lower (or areas with small gradients if 'apply gradient' is selected).
- **Selections:** If a selection has been defined on the geometry, it's possible to apply different size limits to the area of the geometry within the selection. In addition, it's possible to designate the selected part of the geometry boundary as an open boundary for the mesh.

Rectangular

The rectangular mesher is available for polygon geometries only.

- **Element width:** This parameter determines the width of the generated rectangles.
- **Element height:** This parameter determines the height of the generated rectangles.
- **Orientation angle:** This parameter determines the angle at which the rectangles are generated
- **Border policy:** This determines how to handle the generation of elements at borders. There are three options:

- **Overlap:** Rectangles will be created and overlap the border where they are too big to fit.
- **Leave gap:** Rectangles will be created until they cannot fully fit, the remaining area of the selected domain will be left empty.
- **Fill with triangles:** Rectangles will be created until they cannot fully fit, and in the remaining area triangles will be generated. If this option is selected, a width must be specified for the triangular buffer, and size criteria must also be specified (see Triangulate for details).
- **Selections:** If a selection has been defined on the geometry, it's possible to apply different size limits to the area of the geometry within the selection.

Channel

The channel mesher is available for polygon and polyline geometries.

- **Resolution method:** Two options are available for specifying the size of the mesh: *Number of elements* and *Size of elements*. When choosing *Number of Elements*, the user specifies the number of rows and columns in the mesh. When choosing *Size of Elements*, the inputs required are:
 - Optimal length flow direction (default: 40m)
 - Optimal length transverse direction (default: 10m)

A *Buffer width* can be specified when using the Channel mesher with a polyline geometry.

Radial

The radial mesher is available for point geometries only.

- **Radius:** The radius the generated mesh should have.
- **Radius of hole:** A hole can be placed at the center of mesh by giving this parameter a value greater than zero.
- **Fill hole:** This switch will fill a center hole with triangular elements.
- **Element length at mesh border:** This enables the user to specify the element length at the outermost part of the mesh.
- **Slice angle:** This parameter enables the user to create a radial mesh for a portion of a circle.
- **Orientation angle.** This parameter specifies the starting angle for a mesh. The default is zero degrees (12 o'clock)
- **Element type:** Users can choose between four- and three-sided elements.

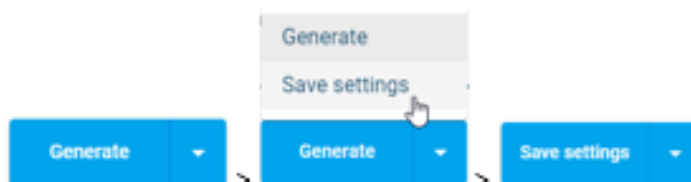
Coastline

The coastline mesher is available for polyline geometries only.

- **Boundary:** The user should select a polygon to define the open boundaries of the coastal mesh..
- **Size criteria:** Users have a choice of two criteria for specifying the minimum size of mesh elements: area or edge length.
- **Maximum element face length (m) or area (m²):** This limit is applied throughout the meshing area.
- **Element length (m) or maximum area (m²) at open boundary:** This constraint is applied along the open boundaries of the mesh.
- **Element length (m) or maximum area (m²) at coastline:** This constraint is applied along the coastline.
- **Buffer:** This parameter maintains the coastline element size specification for a user-specified distance from the coastline.

- **Area of interest:** This enables using a second polygon to define an area of interest within the domain where a different maximum size criterion can be applied.
- **Flip land:** The coastline mesher will automatically locate the mesh in the part of the bisected boundary polygon with the largest area. This switch enables the user to select the reverse.
- **Enable gradual size:** If a different sizing criterion is applied at a mesh border, this switch will enable a gradual transition.
- **Use background density function:** This switch enables use of a variable to control the density of mesh elements. More information is available in Appendix A. This switch activates the following input parameters:
 - **Number of points** (default 10.000): This is the number of mesh nodes to be generated by the density function.
 - **Buffer size:** This parameter excludes a buffer adjacent to the geometry from the densification operation.
 - **Select a variable or no variable:** This field enables selection of variable to control the density of mesh elements. The density of elements will be increased in areas with larger values. It's also possible to use the density function without a variable. If a variable is selected, two additional switches can be applied:
 - **Apply gradient:** This switch uses a gradient instead of actual values to drive densification, with higher density applied in areas with larger gradients.
 - **Invert the density function:** This option will increase the mesh density in areas where variable values are lower (or areas with small gradients if 'apply gradient' is selected).

Once the parameterization is finished, there are two options: save settings - this option will save the current work, but not generate a mesh yet, and Generate - this option will save the current work and immediately start generating a mesh. Per default MB will suggest to Generate, but if you would rather save your settings this can be done by clicking the arrow on the Generate button and then selecting the other option. The button will then switch to the Save settings option and once clicked will perform only a save but not generate a new mesh yet:



Both options will generate a new object, a Mesh Object. This will appear under the header *Meshes* in the main Workspace page.

A Mesh Object can be duplicated, deleted and saved, as well as structurally modified by post-mesh operation methods (see 3.8.1), or updated by changing input data and/or control parameters.

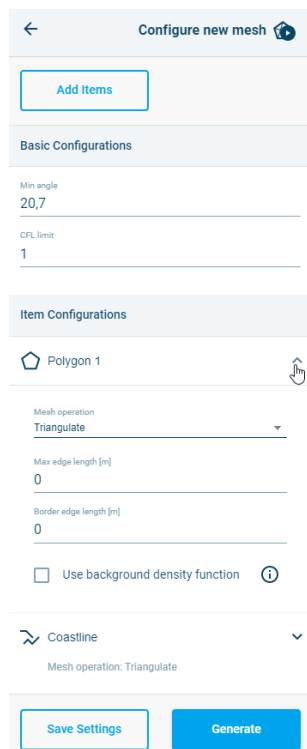




Figure 3.16 Selecting input configuration parameters.

3.7.2 Adding items to a Mesh Object

In order to add any new variable and/or geometry to a Mesh Object, there are two ways to get to the same point:

1. The first option is from the Main Workspace page.

Go to the variable or geometry and click on the  menu button behind the name, then click  Add To Mesh.

An overview will appear with available meshes. Boxes will be ticked and greyed out for meshes that already contain the object.

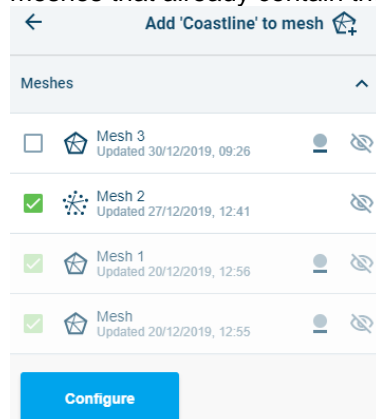



Figure 3.17 Selection of meshes to add the item to.

Tick the boxes for the meshes that you want to add your data to, then click Configure.

Note: adding items will cause Mesh Objects to be remeshed.

2. Alternatively, enter the mesh object (from the main Workspace page, click on the  button behind the mesh under the Meshes header).

Click on tab Configuration at the top of the page, then click the Edit configuration button at the bottom and then click Add items at the top. This will show an overview of available objects. Items that are already a part of the Mesh Object will be ticked and greyed out.

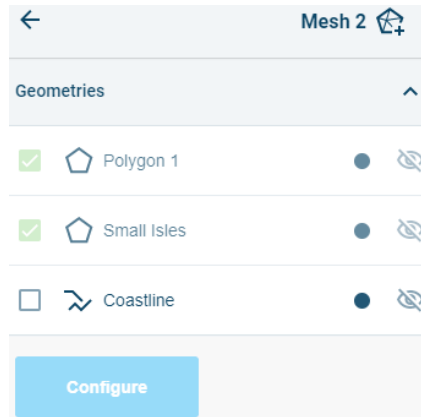



Figure 3.18 Selection of items to add to the mesh.


Tick the boxes for the items that you want to add to the mesh.
Note: adding items will cause Mesh Objects to be remeshed.

3.7.3 Removing items from a Mesh Object

Any variable and/or geometry can be removed from an existing Mesh Object.

Click on the  button behind the mesh under the Meshes header in the main Workspace page to go into that mesh.

You will enter the Overview tab.
 Scroll to the bottom to find the header Mesh items.

Each of these items has a  button behind it. Using it will remove the item from the mesh and cause the mesh object to be refreshed with the new situation.

3.7.4 Interpolating a mesh

Once the mesh has been generated and looks as desired, it can be interpolated if there are Variables available for interpolation. This can be done from inside the mesh settings by clicking the Interpolate tab:

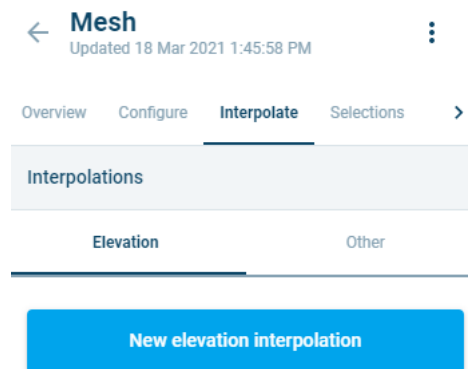


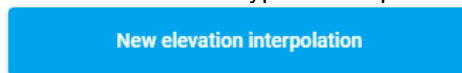
Figure 3.19 Interpolate tab

If there are no available Variables, the user is prompted to add one or more. You can add Variables by editing the workspace (see 3.3.2) and uploading a file with variables, or by drawing your own (see 3.5).

The user can select between two types of interpolation: Elevation (used for applying height and depth values to the mesh) or Other (any other type of interpolation of values defined by the user. The major differences between the two options are as follows:

Elevation	Other
Interpolates to nodes	Interpolates to nodes or elements (choice)
Lets user apply a CFL operation using CFL limit and sea level	No CFL operation
Results come under 'Elevation' variable in the overview	Results come under chosen name by user

Click the tab for the type of interpolation that is desired, then click:



Select the file(s) that should be used in the interpolation by ticking the box(es) in front of them.

Note: MB does not support multiple elevation variables in one location. Instead, a prioritization scheme is used to determine which variable should be used. See below for more information.

Click:



For an *Elevation* interpolation, there will be the option to Apply a CFL operation. When the slider is clicked two more fields will appear:

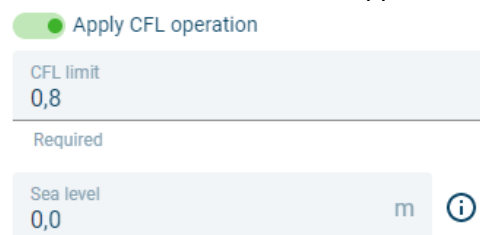


Figure 3.20 Slider for CFL operation active under *Elevation* interpolation type

Based on the *CFL limit* (default 0,8) and the *Sea level* (0m), a Minimum Estimated Timestep is calculated and added to the Mesh Overview after the Interpolation is finished.

For the *Other* interpolation type, two basic settings need to be filled out:

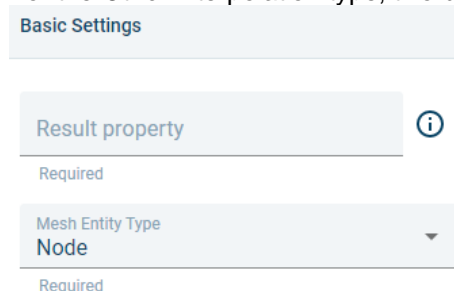


Figure 3.21 Basic settings for the *Other* interpolation type

The *Result property* is the name the property will have once it is interpolated, and by which it can be found in the Mesh Overview. The name can be chosen freely.
 The *Mesh Entity Type* determines whether the values are interpolated to the nodes or to the elements instead.

For both types of interpolations the variable settings are displayed below:



Figure 3.22 Variable settings in interpolation

Click on the Edit button.

This will open up the settings:

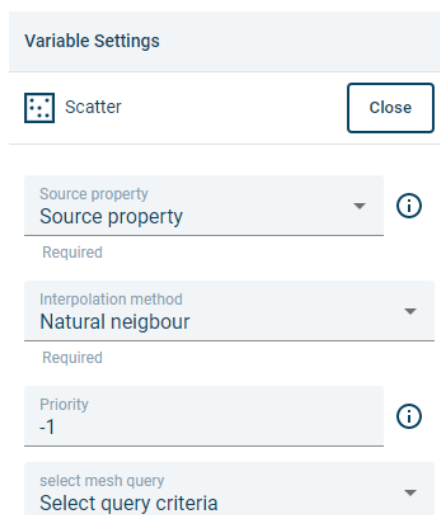


Figure 3.23 Variable settings in edit mode

There are four settings:

- *Source property*: The property that is used to determine the values. Make sure the correct property is selected.
- *Interpolation method*: There are three interpolation methods:
 - Natural neighbour: The values are based on Voronoi tessellation (default).
 - Inverse distance weighting: The values are calculated with a weighted average of the values available at the known points.
 - Linear: Values are determined by a method of curve fitting using linear polynomials.
- *Priority*: If an interpolation uses two or more variables, then the priority setting is used to determine which variable to use in case of overlap. The lowest number establishes the highest priority.

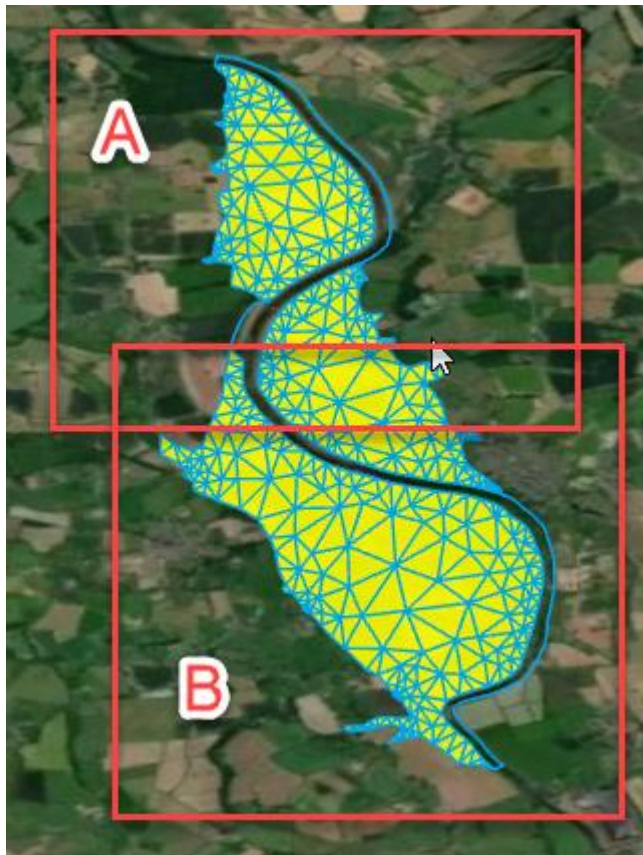


Figure 24 Example of variable prioritization. If A has higher priority, then A will be used to interpolate the mesh in the area where A and B overlap.

- *Select query criteria:* The interpolation can be applied to parts of the mesh, determined by the query criteria. If, for example, the interpolation should only be applied on a certain region or user defined selection area (make the selection on the Geometry), it is possible to select it here.

Note: when using mixed scatter and raster datasets, they cannot overlap. Selections or Regions need to be defined to make sure the interpolation can happen correctly.

Once this is done for the variables that should be interpolated, click:




The interpolation is done in the background. Once it is done, it is possible to view it by clicking the **Elevation** (for the *Elevation* type), Minimal Time Step or named option (for the *Other* type) in the Mesh Analysis, or to go to the 3D mode and view the whole there.

3.7.5 Exclude items from mesh (holes)

Depending on the application domain, holes can have different representations. They either correspond to a physical reality or a conceptual definition. A hole in a mesh is an internal area where elements are missing, thus defining an internal border. Frequent use of holes in meshes is found in urban applications involving the presence of buildings and large human-made structures that act as full barrier to surface flows. Islands are often excluded from a marine

application mesh using such holes. Inland water bodies such as lakes may also be excluded from the mesh and further handled by means of engine specific boundary conditions.

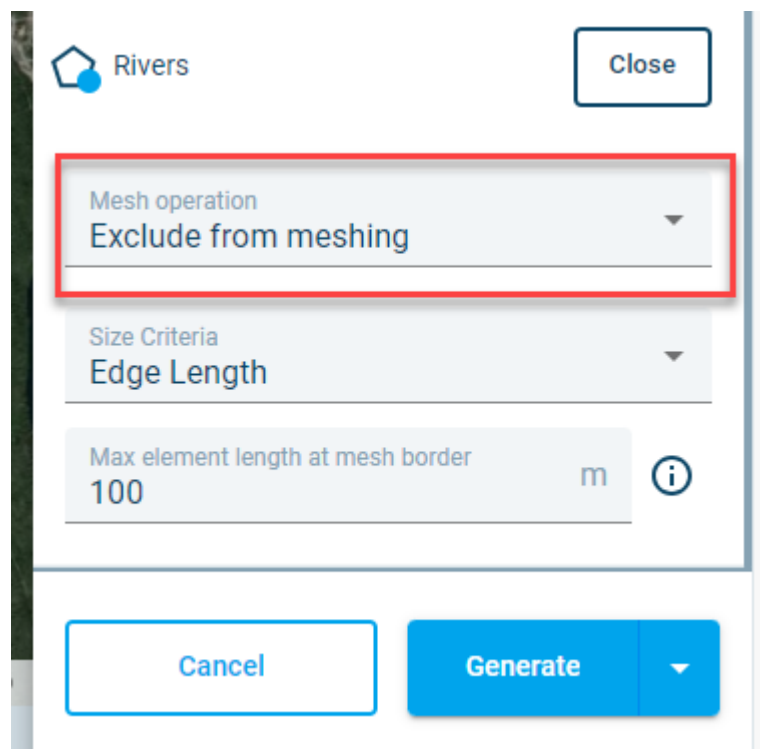
Polygons with holes can be directly imported from shapefiles. Standard polygons can additionally be assigned (or unassigned) the 'Hole' status. This is a typical polygon characteristic that Delaunay meshers can account for during the mesh generation which, when successful, delivers back a mesh body with corresponding physical holes.

Geometries can be excluded from the mesh from inside the Geometries panel. From the main Workspace page, click the  button behind the name to enter the Geometry. On the Overview tab, there is a switch:




When the dot is on the left, it means this item is **not** excluded from the mesh. Click on it to move the dot to the right, and thereby exclude it from the mesh.

Alternatively, it's possible to exclude a geometry in the mesh configuration settings.



3.8 Analysing and improving meshes

When a mesh is produced, MB generates elements and nodes in relation to the domain spaces defined by input geometries. These sub-domains and localized paths geometric definitions are translated into different categories which are assigned to mesh elements and nodes. Polygonal regions are tentatively inherited by mesh elements using *Region* identifiers. Mesh nodes on polygonal paths and constraints are assigned specific IDs.

This data can be used for analysis and is displayed inside the Mesh Object, which (after creating a new mesh) can be found from the main Workspace page when clicking the  button behind the mesh object name.

On the Overview tab that appears in the panel, an area called Mesh Analysis holds this information (see Figure 3.25).

Any of the items (None, Node markers, Element types, Condition no., etc.) that are represented below the Element count, can be clicked to reveal statistics.

The map area will display a Legend on the top right and change visual representation to show the selected option.

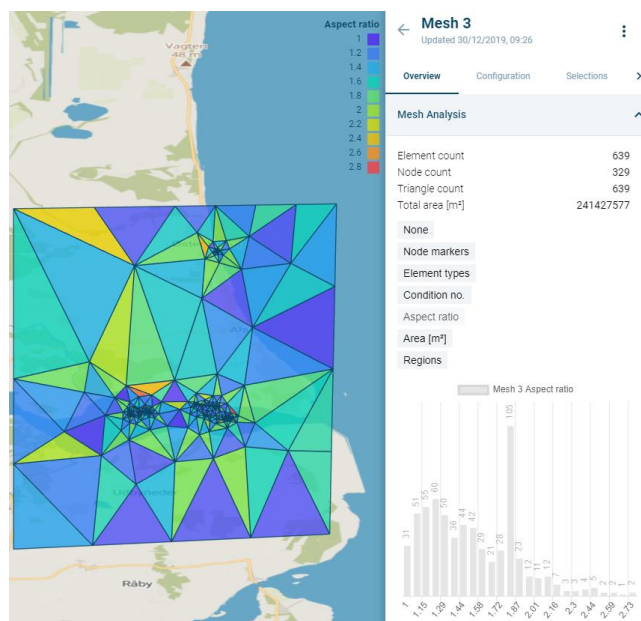



Figure 3.25 Expanded Mesh Analysis with Aspect ratio clicked open.

Mesh information corresponds to nodal groups (*Node markers*) and elemental groups (*Regions, Types*), as well as geometric and quality information (*elemental areas, quality metrics*).

It is possible to improve the mesh by using Mesh post-operations (automated improvements over bigger areas) and through Queries (finding individual points and editing them).

3.8.1 Mesh post-operations

It is possible the Mesh Object could be improved by some generic operations. The MB currently provides the option to apply post-operations to the entire mesh object or on a subset of its elements, which can be selected through a query mechanism.

These queries and post-operations can be found inside the Mesh Object, which (after creating a new mesh) can be found from the main Workspace page when clicking the  button behind the mesh object name.

Once inside the Mesh object, click the  menu at the top right, then select  Operation.

The panel shows the Operations tab, with a dropdown that enables the selection of the desired operation (see Figure 3.26).

Note: When applying an operation to a mesh, a copy is made to which the operation is applied. This copy does not immediately become visible in the map.

Post operations

Mesh 1

Group Name

Post operation
 select post operation ▼

Cancel

Submit operation

Figure 3.26 Operations tab in the Mesh Object.

The list of currently available post-operations:

- *Delete elements*
Deletes the selected subset of elements, leaving holes.
- *Extend mesh*
After the selection of a polygon in the *Extension item ID* field, this operation attempts to create a new mesh from the original mesh and the selected polygon, leaving the selected mesh as much untouched as possible.
The *Snap tolerance* parameter can be used to make gaps disappear between the original mesh and the new addition. The *Element buffer level* determines how many elements deep the original mesh can be changed by the algorithm (1 meaning the first layer of elements can be altered i.e. the area where the extension touches the mesh).
Note: when going from very coarse to very fine elements, the *Element buffer level* can help smooth the mesh elements – of course at the cost of changing more of the original mesh.
- *Quad-dominant*
Attempts to merge adjacent triangles to create quadrangles. This operation generally creates a mixed-element mesh in which the proportion of quadrangles is bigger than the proportion of triangles. It can be viewed as a coarsening operation that minimizes the negative impact on simulation engine's stability that is generally induced by coarsening operations. This minimization of potential coarsening damage relies on the fact that quadrangles are numerically superior to triangles because they allow gradients (referring to the property gradient approximation inherent in PDE solving numerical methods) to vary over the element while triangles only support constant gradients.
- *Refine*
This method will split the targeted elements to increase mesh density. Each quadrangle will convert into 4 quadrangles and each triangle will convert into 3 triangles.
- *Local remesh*
This operation will remesh only the elements determined by the query – leaving the rest of the mesh intact.
With the *Include add-in* slider it is possible to determine a Geometry that should be included, for example to allow houses as holes (these will need to be imported or drawn as a Geometry beforehand and then marked as holes).
The *Element buffer level* field determines determines how many elements deep the original mesh can be changed by the algorithm (1 meaning the first layer of elements can be altered i.e. the area where the queried selection touches the mesh).

Note: it is possible to use a Geometry as a selected area by selecting Geometries as the query location and then picking the desired geometry for the location in the dropdown. This Geometry needs to be created beforehand.

- *Smooth*
Applies a Laplacian smoothing to selected element nodes to improve mesh quality.


After the first dropdown selection is made, a second dropdown appears, allowing for specification of which Mesh elements the operation should be applied to. If a specific area needs to be affected by the operation, it is possible to create a Selection (see 3.8.2 Selections).

- Select the desired post operation.
- Select the desired mesh elements.
- Click Submit.

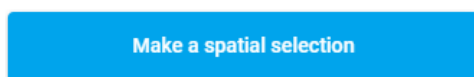
Note: remeshing the Mesh Object with the Generate button from the mesh menu will remove post-operations applied to it and generate the mesh from the chosen Mesh Configuration only.

3.8.2 Mesh selections

Selections are areas of the mesh with a specific meaning to the user. They can, for example, be areas where the mesh should be finer or where elements need to be removed. They can be used as input values in the Mesh post-operations.

They can be found from the main Workspace page when clicking the  button behind the mesh object name and then clicking the tab Selections.

The tab gives an overview of existing Selections and allows the user to draw a new selection.



Click:

Draw the desired area on top of the mesh on the left by clicking with the left mouse button at a starting point, then clicking once for each desired area point and then double click to finish drawing. The result should be highlighted like so:

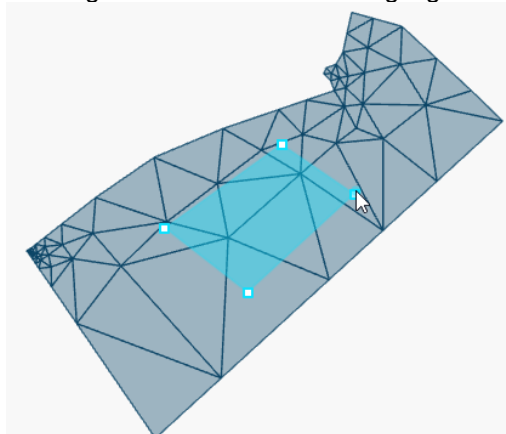


Figure 3.27 Selection on top of a mesh

A name can be specified for the selection, which will show up in dropdown where it can be used. If no name is given, the default name Selection [auto-number] will be applied. Click Create to add the selection.

It will then be available in *Post operations* and *Queries* in the dropdown with desired mesh elements under Selections:

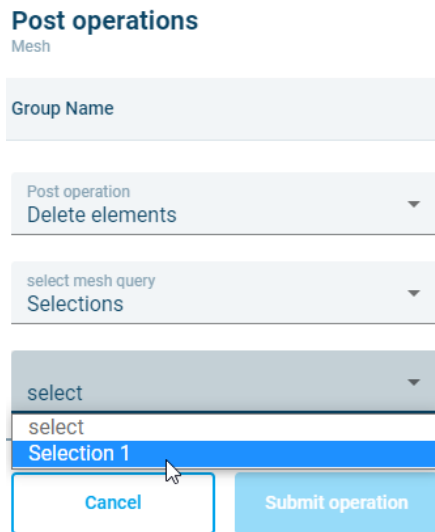



Figure 3.28 A selection used in Mesh Post-Operations

3.8.3 Queries

The queries can be used to quickly visualise elements of a mesh or to find specific elements (and edit them if need be).

They can be found from the main Workspace page when clicking the  button behind the mesh object name; behind the tab Selections is an option to scroll further right. Click on the arrow pointing right:

Selections 

The tab Query is revealed.
Click on the tab.

A panel with a dropdown appears, allowing for the selection of the desired query criteria.

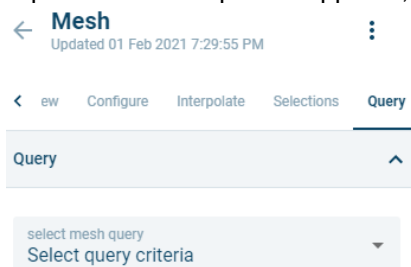


Figure 3.29 The Query panel

Click on the desired query.

A table appears below the selected query. In some cases it is possible to select a range on top of it, in case the default isn't specific enough. The table shows a top 10 of elements based on the query.

select mesh query
Area [m²]

min
≥ 13.1

max
≤ 6135966

Top 10























1	13.126661142713932		
2	39.02021314994165		
3	168.7909676164513		
4	188.33991962039		
5	287.8891891282384		
6	343.7402203253309		
7	575.7808088432612		
8	741.7222853928173		
9	829.8739965808422		
10	1098.97460807128		

Figure 3.30 Overview of smallest elements in the mesh


It is possible to zoom to the mesh element in the table by clicking .

In order to change a mesh element:

Click  and click with the left mouse button on a node on the map. The node and its connecting nodes will light up.

Drag the node to desired position, making sure not to cross the highlighted boundaries.




Click  to put the node in its new position. The mesh will be redrawn.

Note: remeshing the Mesh Object will remove changes applied to it and generate the mesh from the chosen Mesh Configuration only.

3.9 Exporting data

It is possible to get data out that was created inside the MB, for use in other applications.

To export data: click on the  button in the rightmost menu bar. This will open up the Export data panel. At the top of the panel is a dropdown listing the item types that can be exported: Meshes, Geometries and Variables.

Select the desired item type from the dropdown.

Tick the boxes in front of the item(s) that you want to export:



Click the button at the bottom:

Export mesh

Files will be placed in the root folder of your project in Data Admin in the Cloud. They can be downloaded from there.



Meshes can be exported in two different formats:

1. The MIKE Zero *.mesh ASCII format
2. Kitware's VTU format

4 Step-by-step mesh generation example

This section briefly shows how to create a mesh with example data. We are using the polygonal description of a part of River Severn in the United Kingdom, close to Upton Severn city.

The desired mesh deliverable corresponds to a quality, mixed-elements mesh designed for flood simulations in which quadrangles will cover the river space and triangles occupy laterally connected zones. The data consists of a central polygon that maps the river section where a smooth curvilinear quadrangular mesh is wanted. Three additional polygons plug to the sides of the river polygon, defining riverside flood plains where triangulation is performed.

To download the dataset, log in to MB, click  at the top right and select  [Download example data](#).

A file of 17.5 MB will be downloaded to the default folder on your computer. This file contains 3 sets of example data, of which we will be using only River Severn. You are free to try the other data sets to practice further.

You will need to create a project in the MIKE Data Admin application, and upload the example files, as per the instructions in chapter 3.2.1. You can then continue from the MB:

1. **Selecting the Project**
Select the Project you have created from the Severn River data
2. **Creating a Workspace**
Select all data records and create a Workspace
3. **Triangular meshing**
As a preliminary meshing event, a pure triangulation is performed. In order to better constrain the triangulation in the river polygon, its centreline is calculated in a pre-meshing step. The obtained polylines record is then added to the list of geometries to be meshed:
 - a. Enter the geometry panel of “River” (see Section 0)
 - b. Create its medial-axis by drawing a polyline (see Section 3.4)
 - c. Go back to the main page and create a new mesh (see Section 3.7)
 - d. Select all geometries, including the newly created medial axis
 - e. Generate the mesh (thus using default parameters)
4. **Analysing the mesh**
 - a. Enter the generated mesh panel (see Section 3.7.5)
 - b. Visualize its quality metrics
 - c. You may want to perform queries and launch post-operations like mesh smoothing (see Section 3.8.1)
 - d. You may also want to change the default settings and do remeshing
5. **Creating a mixed triangular / quadrangular mesh**
 - a. Enter the triangular mesh object and duplicate it (see Section 3.4.3)
 - b. Exit the mesh object panel and enter the duplicated one
 - c. Go to Configuration tab and click “Edit configuration” button
 - d. Change the river polygon mesher to “*Quadrangulate*” (see Section 3.7.1)
 - e. Change default number of rows and columns if wanted
 - f. Regenerate the mesh
6. **Saving meshes**
Follow Section 3.9 to export one or more meshes

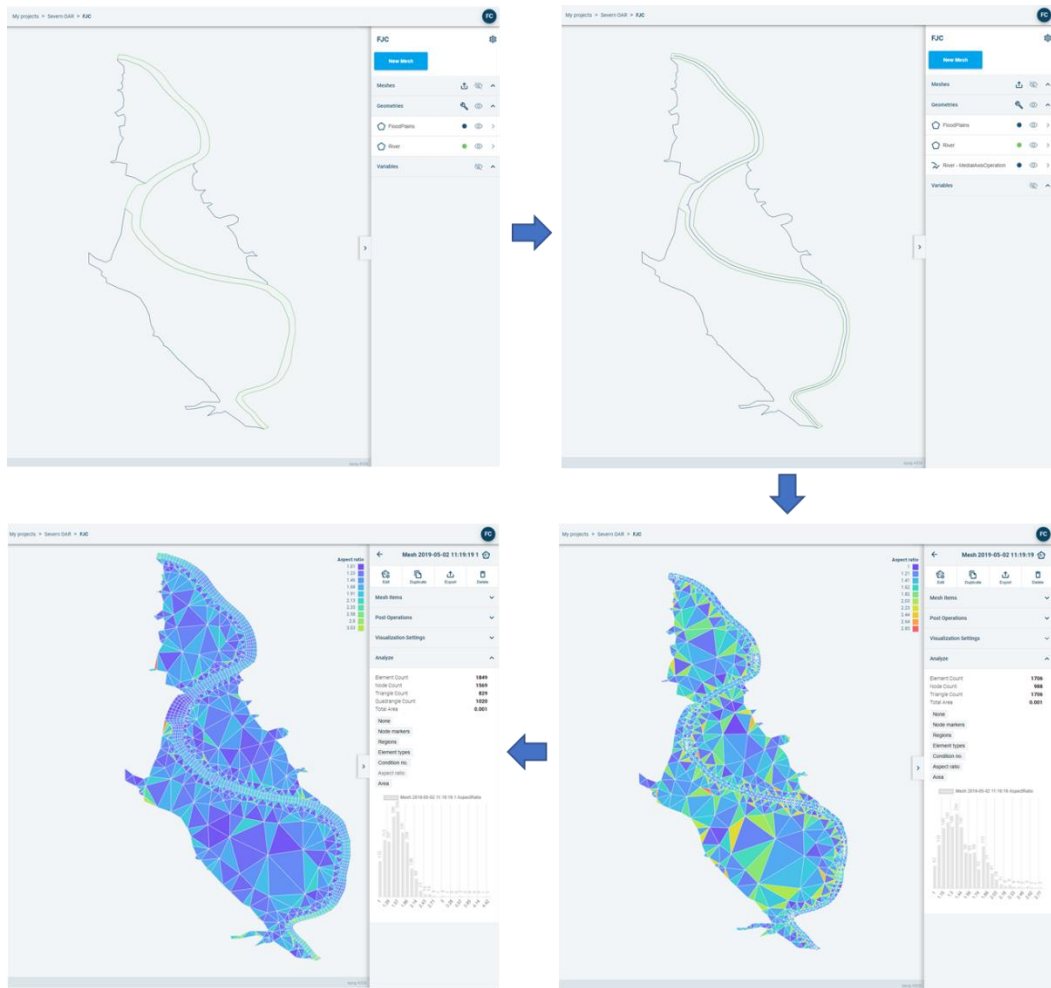


Figure 4.1 Generating two variants of a mesh step by step.

5 Problem solving

Since MB is an online service, always check if there are notifications about maintenance on the MIKE Cloud Homepage. If this is not the case and you have repeatedly experienced a problem, this chapter describes how to deal with problems.

5.1 File won't upload

When you can't get a file to upload (i.e. it won't appear in the MIKE Data Admin or you receive an error warning during the upload) please contact Customer Service. Please include the file you have tried to upload and the error description that you got. A date and time are also convenient for analysis.

5.2 File doesn't work as expected

When you are trying to use data from a file that you uploaded and it doesn't work as expected in MB, check the following:

- Is the file of a supported format (see 3.2.1)?
- Did you upload it with the correct settings for the file format?
- Are you using only one projection system between multiple files (see 3.2.3)?

If these things are in order and you still experience issues, please contact Customer Service. Please include the file you have tried to upload and the error description that you got, as well as a description of when during the process you got it. A date and time are also convenient for analysis.

5.3 Maps won't load / visual inconsistency

When map parts won't load it is possible the system was unable to retrieve them at a certain point in time; the browser (Chrome or Edge) sometimes 'remembers' this for a while (caching). This is normally used so previously requested data can be reused, but sometimes this leads to problems.

When you encounter a problem with caching, please try pressing CTRL and F5 at the same time. This is a forced refresh for the browser, and it should refresh all things it currently holds in cache as well (which pressing F5 alone won't always do).

This fix should also help with other visual inconsistencies. If these you still experience issues, please contact Customer Service. Please include a screenshot (F12 and then paste in your e-mail) of the issue and a description of what you were trying to do and what went wrong. A date and time are also convenient for analysis.

A Appendix: Generating meshes controlled by a background density function

One major difficulty frequently encountered in unstructured meshing is the ability to control the variations in mesh density (i.e., the spatial variations in elemental sizes) over the entire domain.

A powerful approach is therefore proposed to achieve such a task. The method relies on the presence of a user-defined (mesh) density function in the form of a raster variable. This variable can be arbitrary, while expected to be positive definite. The content is directly taken as expected mesh density value in the 2D (x, y) Euclidean space. Optionally, the gradient of the submitted variable can be taken as density function.

A.1 Method description

The overall principle is quite simple and relies on a *Centroidal Voronoi Tessellation* (CVT) technique whose aim is to create a point cloud equal to centers of Voronoi cells, i.e. the dual graph of a Delaunay-conform triangulation.

The minimization of an energy functional will typically generate point clouds that satisfy Voronoi constraints, i.e., the created points are centres of Voronoi cells. Consequently, such a point cloud can eventually be converted into a 2D or 3D Delaunay-conform triangulation by minimally connecting the Voronoi cell centers.

Furthermore, the approach naturally allows a smooth and continuous control on the spatial variations of mesh densities when adding a rejection principle that consumes a background density function.

A CVT can be summarized as follows. Let $|\cdot|$ be the Euclidean norm in $\mathbb{R}^{d=2,3}$, $\Omega \subset \mathbb{R}^d$ be an open set, and $\{\mathbf{x}_i\}_{i=1}^n$ be a set of n points in Ω . The Voronoi tessellation (or Voronoi diagram) of Ω is the set $\{V_i\}_{i=1}^n$ defined as:

$$V_i = \{\bar{\mathbf{y}} \in \Omega \mid |\bar{\mathbf{y}} - \bar{\mathbf{x}}_i| < |\bar{\mathbf{y}} - \bar{\mathbf{x}}_j| \quad \forall j = 1, \dots, n \text{ and } j \neq i\}, \quad i = 1, \dots, n \quad (5.1)$$

$$V_i \cap V_j = \emptyset \quad \text{for } i \neq j \quad \text{and} \quad \bigcup_{i=1}^n \bar{V}_i = \bar{\Omega} \quad (5.2)$$

The set $\{\mathbf{x}_i\}_{i=1}^n$ is referred to as generating points or generators, while each V_i corresponds to a *Voronoi region* (or *cell*) corresponding to $\bar{\mathbf{x}}_i$. Given a density function $\rho(\bar{\mathbf{x}}) \geq 0$ defined for all $\bar{\mathbf{x}} = (x, y, z) \in \Omega$, then for each Voronoi region V_i the *mass centroid* or *centre of mass* $\bar{\mathbf{v}}_i$ of V_i is defined by:

$$\bar{\mathbf{v}}_i = \int_{V_i} \bar{\mathbf{x}} \rho(\bar{\mathbf{x}}) dV_i / \int_{V_i} \rho(\bar{\mathbf{x}}) dV_i, \quad i = 1, \dots, n \quad (5.3)$$

A tessellation as defined by this set of equations is called a CVT if and only if $\bar{\mathbf{v}}_i = \bar{\mathbf{x}}_i$, $i = 1, \dots, n$. This is equivalent to saying that all points $\bar{\mathbf{x}}_i$ act as generators associated with the Voronoi regions V_i and are the mass centroids of these regions.

Now let define the following energy functional:

$$\Xi(\{\mathbf{x}_i, V_i\}_{i=1}^n) = \sum_{i=1}^n \int_{V_i} \rho(\mathbf{y}) |\mathbf{y} - \mathbf{x}_i|^2 d\mathbf{y} \quad (5.4)$$

A necessary condition for \mathcal{E} to be minimized is that $\{x_i, V_i\}_{i=1}^n$ is a CVT of Ω . The control on the desired mesh spatial density $\rho(\vec{x})$ is achieved by applying a rejection principle during a random generation of the point clouds.

A.2 Pseudo-algorithm

The following pseudo-algorithm is for the 2D (x, y) Euclidean space. The 3rd dimension follows the same principles and is adding no further complexity. The input polygon P and density function ρ can be mapped into the unit square domain for convenience, $(x, y) \rightarrow (u, v) \in [0; 1]$. The generation domain becomes $\Omega = [0; 1] \times [0; 1]$ masked by P .

```

0. Define how many points to generate as mesh nodes:  $n$ 
   Define a number of sampling points:  $m < n$ 

1. Generate the  $n$  points  $\vec{v}$  randomly

2. Generate the  $m$  sampling points  $\vec{s}$  using a Monte-Carlo method with density
   function  $\rho(\vec{x}) \geq 0$  as driver of the rejection principle:

   Generate a uniform random number  $u \sim U(0;1) = x_p$ 
   Generate a uniform random number  $v \sim U(0;1) = y_p$ 
   if ( $p = (u, v)$  in  $\Omega$ ) {
     Generate a uniform random number  $k \sim U(0;1)$ 
     if ( $k < \rho(\vec{x})$ )   accept  $p$ 
     else                reject  $p$  and retry
     Add  $p$  to sampling list  $\vec{s}$ 
     Stop if  $m$  is reached
   }
   else
     reject  $p$  and retry

3. Define the Voronoi region of  $v_{i=1,m}$  gathering together in  $\vec{z}$  the closest
   points among the  $n$  Voronoi points:

   if ( $\vec{z} \neq \emptyset$ ) {
     Compute the average  $\vec{y}$  of the set  $\vec{z}$  and update  $\vec{v}$ :  $\vec{v}_i \leftarrow \frac{(\alpha_1 c_i + \beta_1) \vec{v}_i + (\alpha_2 c_i + \beta_2) \vec{y}_i}{c_i + 1}$ 
      $c_i \leftarrow c_i + 1$ 
   }

4. Go to 2 or stop iterations if the new Voronoi points meet some convergence
   criterion and/or if the maximum allowed number of iterations is overflown

5. Connect the Voronoi cell centers to finalize the mesh using a constrained
   Delaunay mesher, with or without refinement relationships.

```

Remarks:

- The update of the Voronoi cells position starts with initial condition $c_0 = 1$;
- The set of coefficients $\{\alpha_i, \beta_i\}$ must satisfy the constraints $\alpha_1 + \alpha_2 = 1$, $\beta_1 + \beta_2 = 1$, $\alpha_2 > 0$ and $\beta_2 > 0$;
- Therefore both α_1 and β_1 may be negative, so the ensemble $\{\alpha_i, \beta_i\}$ defines under- and over-relaxation updating methods;
- The case $\alpha_1 = \beta_1 = 0$, $\alpha_2 = \beta_2 = 1$ yields $\vec{v}_i = \vec{y}_i$, i.e. the average of the points in \vec{z} . Since the points in \vec{z} are randomly generated in the Voronoi region corresponding to \vec{v}_i , \vec{y}_i can be regarded as a probabilistic approximation to the centroid of V_i , so that this configuration is like a probabilistic version of a Lloyd's method.

A.3 Examples

In this example, the English Channel is meshed. The polygonal data corresponds to a closed multi-polygon object with holes (defining islands) espousing the coastlines of France and of the UK. Bathymetry data is used to formulate a mesh density function in which the correlation engaged looks at the bathymetry variations, in form of the absolute value of bathymetry gradient, $\rho(x, y) = |\nabla z(x, y)|$. Consequently, the higher the bathymetry local variation the denser the mesh is expected.

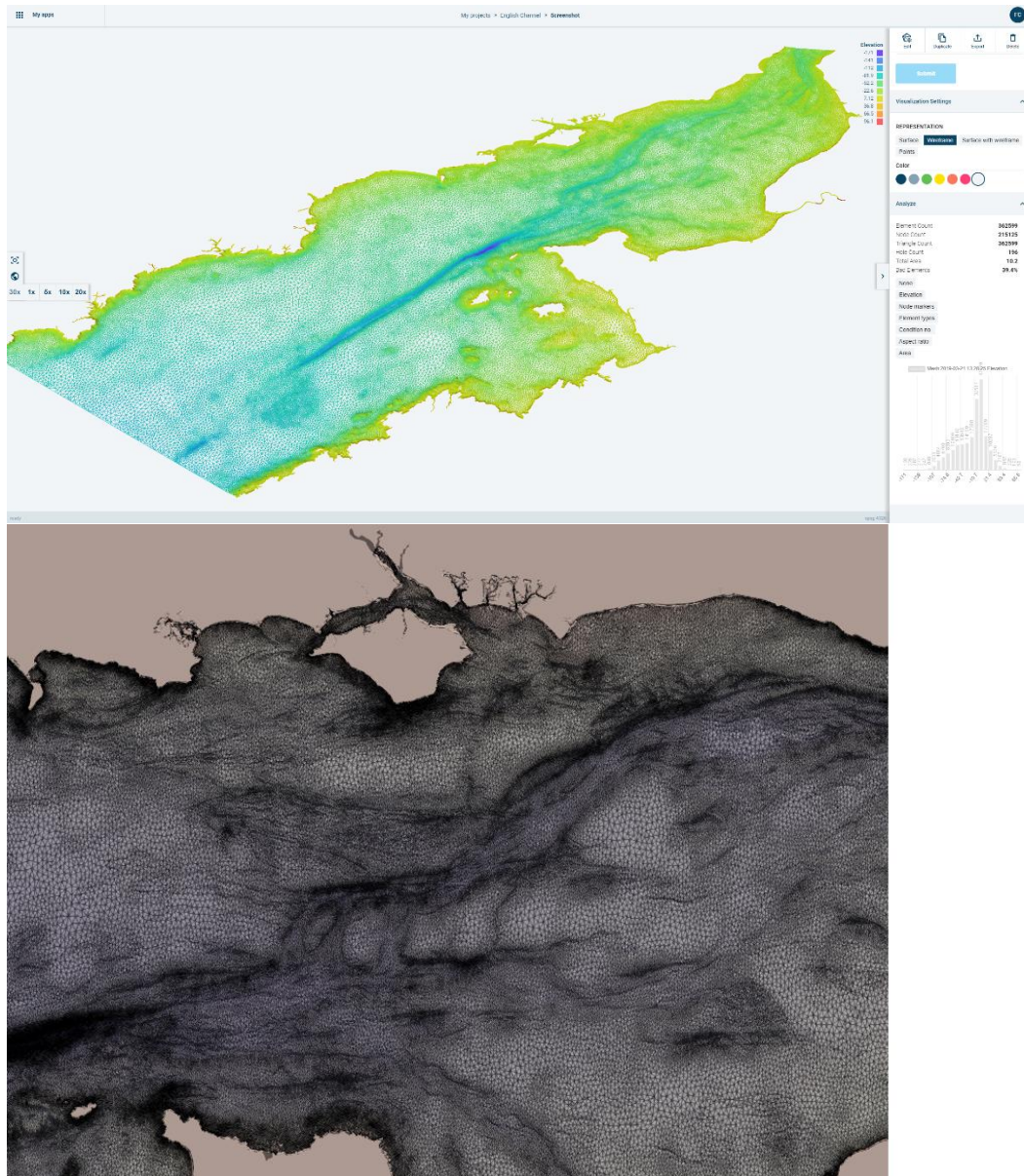


Figure 5.1 Density function aided meshing results. Top: Mesh density correlated by bathymetry gradient (the coloured variable is nodal elevation); Bottom: Close-up on the mesh revealing the smooth densifications and relaxations of mesh density in relation to bathymetry gradients.

B Appendix: Libraries used

Not all functionality in the MB is built by DHI from the ground up. For some things, we use third party software.

The following libraries are used for the geometric calculations in the Mesh-Algorithms package:

- DotSpatial.Projections by DotSpatial Team
- NetTopologySuite by NetTopologySuite Team
- ClipperLib by Angus Johnson
- Triangle.NET by Christian Woltering
- nn (Natural Neighbours interpolation) by Pavel Sakov
- CSharp-IDW (Inverse Distance Weighting Interpolation) by Henrik Alfke

The following libraries are used for visualisation and other tasks in the front-end:

- Material-UI by MIT
- Maps by Mapbox (mapbox.com)
- Mapshaper (mapshaper.org)
- ol.js (openlayers.org)
- Proj4 (proj4js.org)
- React (reactjs.org)
- Redux (reduct.js.org)
- Turf (turfjs.org)
- vtk.js by Kitware (kitware.com)

The following source is used:

- <https://www.openstreetmap.org>

C Appendix: Non-supported projection systems

The following 29 projection systems are currently not supported by the MIKE Cloud Platform:

5516: S-JTSK/05 / Modified Krovak East North
5515: S-JTSK/05 / Modified Krovak
5525: S-JTSK/05 (Ferro) / Modified Krovak East North
5224: S-JTSK/05 (Ferro) / Modified Krovak
5017: Lisbon 1890 / Portugal Bonne New
3993: Guam 1963 / Guam SPCS
3295: Guam 1963 / Yap Islands
32700: WGS 84 / UTM grid system (southern hemisphere)
32600: WGS 84 / UTM grid system (northern hemisphere)
3173: fk89 / Faroe Lambert FK89
3145: ETRS89 / Faroe Lambert
3144: FD54 / Faroe Lambert
9098: Vanua Levu 1915 / Vanua Levu Grid
3053: Hjørsey 1955 / Lambert 1955
3052: Reykjavik 1900 / Lambert 1900
2986: Perroud 1950 / Terre Adelie Polar Stereographic
2985: Petrels 1972 / Terre Adelie Polar Stereographic
2963: Lisbon 1890 (Lisbon) / Portugal Bonne
2307: Qornoq 1927 / Greenland zone 8 east
2306: Qornoq 1927 / Greenland zone 7 west
2305: Qornoq 1927 / Greenland zone 6 west
2304: Qornoq 1927 / Greenland zone 5 west
2303: Qornoq 1927 / Greenland zone 4 west
2301: Qornoq 1927 / Greenland zone 3 west
2299: Qornoq 1927 / Greenland zone 2 west
2296: Ammassalik 1958 / Greenland zone 7 east
2221: Scoresbysund 1952 / Greenland zone 6 east
2218: Scoresbysund 1952 / Greenland zone 5 east