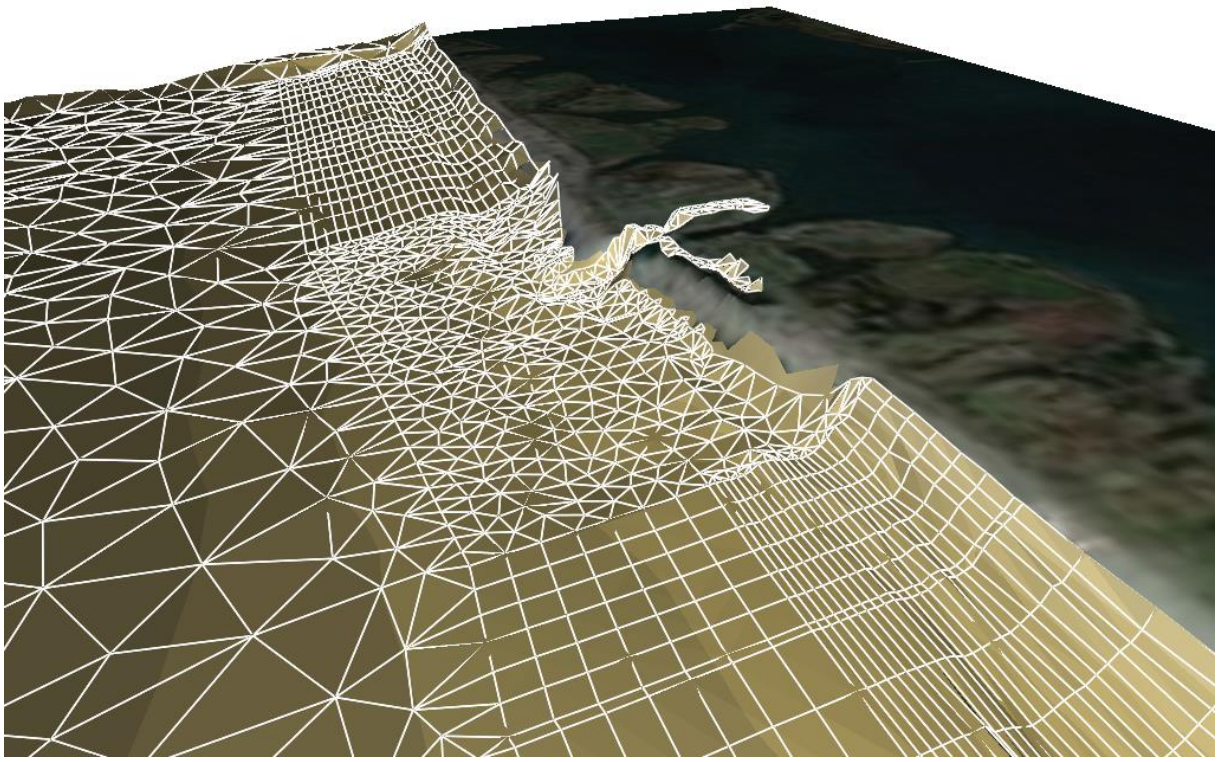


MIKE Zero

Mesh Generator

Step-by-step training guide

DHI A/S headquarters
Agern Allé 5
DK-2970 Hørsholm
Denmark

+45 4516 9200 Telephone
+45 4516 9333 Support
+45 4516 9292 Telefax

mike@dhigroup.com
www.mikepoweredbydhi.com

CONTENTS

MIKE Zero
Mesh Generator
Step-by-step training guide

1 Introduction 1
1.1 Background 1
1.2 Objective 1
1.3 General Considerations before Creating a Computational Mesh 2
1.3.1 Courant number 2
1.4 Remarks 3
1.5 Concepts 4
1.5.1 Arc properties 4
1.6 Mesh File 6

2 Bathymetry Data 9
2.1 Computational Grid 9
2.2 Water Depths 9
2.3 Boundary Data 9
2.4 Background Images 10

3 Initial Computational Mesh 11
3.1 Creating the Workspace 11
3.2 Digitise Model Boundary 16
3.3 Import Model Boundaries 17
3.4 Specification of Domain 18
3.4.1 Editing land boundary 18
3.4.2 Adjusting the boundary data into a domain that can be triangulated 20
3.5 Specification of Boundaries 21
3.6 Mesh Generation 23
3.7 Analysis 24
3.8 Smoothing the Mesh 26

4 Initial Bathymetry 29
4.1 Importing Scatter Data 29
4.2 Interpolating Scatter Data 30
4.3 Analysing the Mesh 31
4.4 Exporting the Mesh 31
4.5 Viewing the Mesh 32
4.6 Next Steps 34

5 Mesh for Regional Wave Simulation 35
5.1 Modifying the Land Boundary 35
5.2 Increasing Resolution 37

5.2.1 Simple refinement of mesh 37
5.2.2 Using smaller maximum element areas 38
5.3 Interpolating Bathymetry 38
5.4 Refine Mesh in Shallow Areas 41
5.5 Influence of Mesh Resolution on Simulation Results 43

6 Mesh for Local Wave Simulation 45
6.1 Available Data 45
6.2 Modifying the Land Boundary 46
6.3 Using Polygons in Mesh Generation 47
6.4 Prioritise Scatter Data 50
6.5 Updating Mesh Bathymetry Using New Scatter Data 53
6.6 Influence of Bathymetry on Simulation Results 56

7 Mesh for Longshore Current Simulation 59
7.1 Defining Polygons for Customisation 59
7.1.1 Defining quadrangular elements 60
7.1.2 Interpolating bathymetry 62
7.2 Neighbouring Quadrangular Element Areas 67
7.2.1 Two adjacent areas 68
7.2.2 Complex combination of quadrangular areas 71

8 Impact of Tidal Flow 77
8.1 Creating the Land Boundary 78
8.1.1 Redistribute vertices on arc 78
8.2 Specification of Domain 82
8.3 Using Break Lines 84
8.4 Increasing the Model Domain 87
8.5 Simulation Results 90

9 Phase II of Previous Investigation 93
9.1 Modifying Existing Mesh File Directly 95
9.1.1 Moving mesh nodes 95
9.1.2 Re-triangulate selected region 97
9.1.3 Adding node points 98
9.1.4 Deleting mesh nodes 99
9.1.5 Interpolating bathymetry 100
9.1.6 Modifying properties of single mesh node 101
9.1.7 Evaluating change 102
9.2 Use Re-established .mdf File for Existing Mesh 102
9.2.1 Import new boundary 103
9.2.2 Extract node point data from mesh 105
9.2.3 Create mesh 107
9.2.4 Interpolate bathymetry 107

1 Introduction

This Step-by-step training guide relates to the bathymetry around Thorsminde harbour on the west coast of Denmark.

Figure 1.1 Thorsminde, Denmark

Setting up a mesh includes appropriate selection of the area to be modelled, adequate resolution of the bathymetry, flow, wind and wave fields under consideration and definition of codes for open and land boundaries. Furthermore, the resolution in the geographical space must also be selected with respect to stability considerations.

1.1 Background

Thorsminde Harbour experienced problems with siltation in the harbour entrance. Some investigations were therefore made to assess the problems and different layouts were tested. As a result, a new harbour layout was carried out resulting in lower maintenance costs.

1.2 Objective

The objective of this Step-by-step training guide is to use the Mesh Generator to create various meshes, each designed to apply for a special modelling task.

The bathymetry mesh is created from scratch and optimised to a satisfactory level.

Attempts have been made to make this exercise as realistic as possible although some short cuts have been made with respect to the data input. This mainly relates to quality assurance and pre-processing of raw data to bring it into a format readily accepted by the MIKE Zero software. Depending on the amount and quality of the data sets this can be a

tedious, time consuming but indispensable process. For this example guide the ‘raw’ data has been provided as standard ASCII text files and encrypted MIKE C-MAP files.

The files used in this Step-by-step training guide are a part of the installation. You can install the examples from the MIKE Zero start page, by selecting the folder

MIKE_ZERO/MeshEdit/Torsminde

Please note that all future references made in this Step-by-step guide to files in the examples are made relative to the main folder holding the example.

User Guides and Manuals can be accessed via the MIKE Zero Documentation Index in the start menu.

1.3 General Considerations before Creating a Computational Mesh

The resulting computational grid file in Flow Model FM is an ASCII file with extension (.mesh) that includes information of the geographical position and water depth at each node point in the mesh. The file also includes information about the node connectivity of the triangular and quadrangular elements. All the specifications for generating the mesh file are saved in a Mesh Definition File with extension (.mdf), which can be modified and re-used.

The bathymetry and mesh file should

1. describe the water depths in the model area,
2. allow model results with a desired accuracy, and
3. give model simulation times acceptable to the user.

To obtain this, you should aim at a mesh:

- with triangles without small angles (the perfect mesh has equilateral triangles)
- with smooth boundaries
- with high resolutions in areas of special interest
- based on valid xyz data using the same chart datum

Large angles and high resolutions in a mesh are contradicting with the need for short simulation times, so the modeller must compromise his choice of triangulation between these two factors.

In areas with a pre-dominant flow direction it may be feasible to apply a quadrangular mesh in order to reflect the flow pattern more accurately.

1.3.1 Courant number

The Courant–Friedrichs–Lewy condition (CFL condition) is a necessary condition for convergence while solving certain partial differential equations. It arises when explicit time-marching schemes are used for the numerical solution. As a consequence, the time step must be less than a certain time in many explicit time-marching computer simulations; otherwise the simulation will produce wildly incorrect results.

The basic CFL number for wave simulations has the following form:

$$CFL_{SW} = c \cdot \frac{\Delta t}{\Delta x} \quad (1.1)$$

Where c is the wave celerity, Δt is the time step and Δx is the spatial resolution.

For flow modelling the CFL number is influenced explicitly by the water depth as well:

$$CFL_{HD} = \left(\sqrt{(g \cdot h)} + |u|\right) \cdot \frac{\Delta t}{\Delta x} + \left(\sqrt{(g \cdot h)} + |v|\right) \cdot \frac{\Delta t}{\Delta x} \quad (1.2)$$

Where g is gravity, h is the water depth, u and v are the velocity components in the x- and y-direction, Δt is the time step and Δx is the spatial resolution.

The resolution of the mesh, combined with the water depths and chosen time-step governs the Courant numbers in a model set-up. The maximum Courant number shall be less than 1.0. So the simulation times dependency on the triangulation of the mesh, relates not only to the number of nodes in the mesh, but also the resulting Courant numbers. As a result of this, the effect on simulation time of a fine resolution at deep water can be relatively high compared to a high resolution at shallow water.

1.4 Remarks

The time spent on creating a reliable mesh before starting the actual simulations is well spent. However it should be anticipated that the first simulations may give rise to the need of further modifications of the mesh as the results are analysed and may found lacking in detail.

Before finalising a project you should anticipate several re-runs of the standard mesh commands: generate, smooth and interpolate.

Beware that the programme will keep the latest temporary setups in memory to allow for the ‘undo’ functionality. The maximum number of saved setups is defined in the Options dialogue.

Please note that for large setups you can lower the level and thus reduce the memory needed for holding the undo buffers.

1.5 Concepts

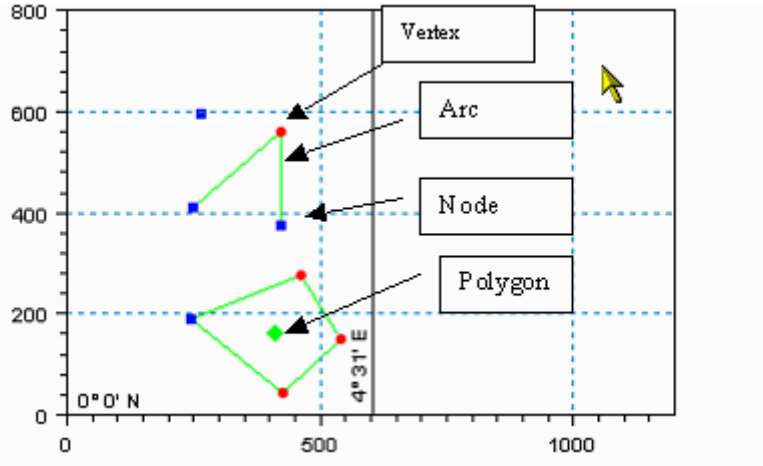


Figure 1.2 Concepts in the Mesh Generator

The concepts in the mesh generator (cf. Figure 1.2) can be explained as follows:

- A *node* defines the start and end point of an arc. A closed arc will only have one node while an open arc will have two nodes.
- A *vertex* is an internal point along an arc.
- An *arc* is a number of line segments composed of zero or more vertices and one or two nodes.
- A *polygon* is a closed loop composed of one closed arc, or two or more open arcs. A polygon is not explicitly defined by the user - it is automatically generated when closing an arc or a set of arcs.

1.5.1 Arc properties

The arc attribute value identifies where to generate individual boundaries in the mesh. A boundary may be one of two types:

- Land boundary, i.e. a boundary line that separates land from water. When you use the mesh in a simulation, the computational engine implicitly defines a zero velocity boundary condition perpendicular to the arc
- Open boundary, i.e. a boundary line where you when setting up a simulation shall prescribe the boundary condition

The first and last node of an arc is defined as the positive direction of rotation around the model area, i.e. counter clockwise

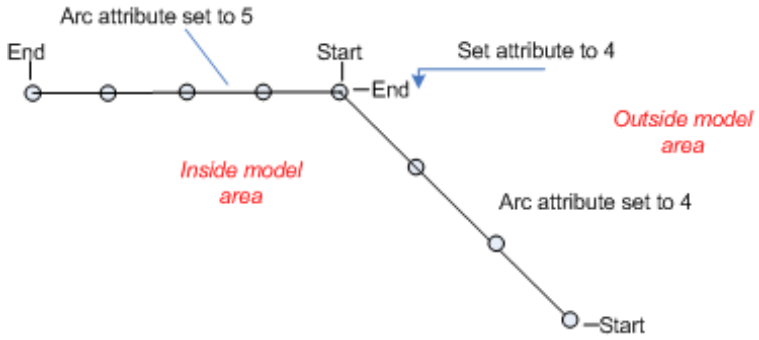
A node – which by definition is the start and end point of the arc – typically belongs to two arcs. This implies that you have to pay special attention to the attribute values you give the start and end nodes.

Depending on how the arcs connect to each other you shall follow one of the following two rules:

1. Connection of two open boundaries
2. Connection of an open and a closed boundary

Connection of two open boundaries

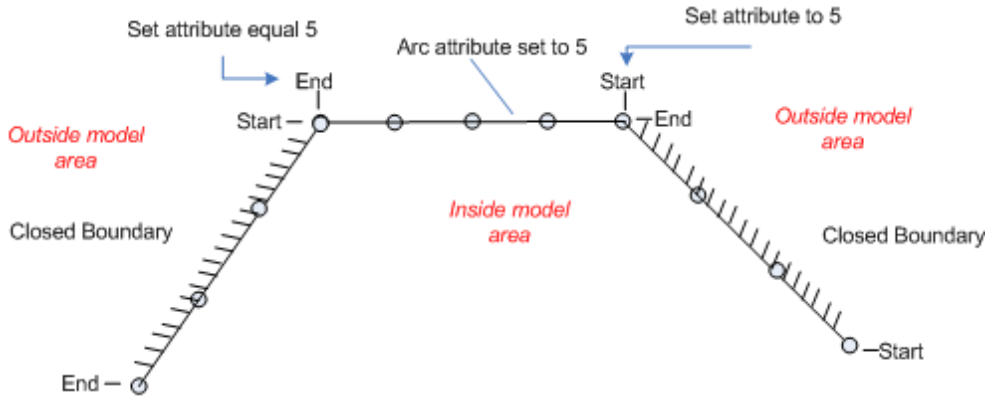
In this case you have to ensure that the end node of the arc is given the same attribute value as the arc itself. This is illustrated below.



Connection of an open and a closed boundary

In this case you have to ensure that the node that connects the open and closed boundaries is given the attribute value corresponding to the attribute value of the open boundary. This is irrespective of the node being the first or last node along the open boundary arc.

The scenario is illustrated below.



Note: During export of a generated mesh, all arc and node attributes with a value of 0 are converted to 1. An attribute value of 1 indicates land. You shall therefore always use a value of 2 or higher as attribute value. Use the same attribute value for boundary nodes and arcs that shall be exposed to the same boundary conditions.

1.6 Mesh File

The mesh file has the .mesh extension and is a text/plain file. It basically lists the node and a simplified element table in text format using spaces as separators.

A mesh file has the following form:

Header line: The first line contains two integers expressing the type and unit of the bathymetry data, and the number of nodes followed by a string defining the projection. (Note: a mesh generated prior to Release 2012 does not contain the type and unit information. The model will then assume the unit to match the information contained in the map projection).

Node lines: As many lines as stated in the header line. Each line contains information for one node on the form:

- The node point Id
- The x coordinate (Easting or Longitude)
- The y coordinate (Northing or Latitude)
- The z value (bathymetry)
- The node type (0 for water, 1 for land and above 1 for all other boundaries)

Element header line: One line containing 3 numbers:

- The number of elements
- The maximum number of nodes per element
- Internal code for mesh type (21 for purely triangular elements, 25 if quadrangular elements exists)

Element lines: As many lines as stated in the element header line. Each line contains information for one element on the form:

- The element Id
- Id for node_1
- Id for node_2
- Id for node_3
- (Id for node_4)

Each line contains at most as many nodes as stated in the element header line. If an element has fewer nodes than the maximum number of nodes per element stated in the element header line, the remaining node Id value is 0.

An example of a mesh file is given in Figure 1.3.

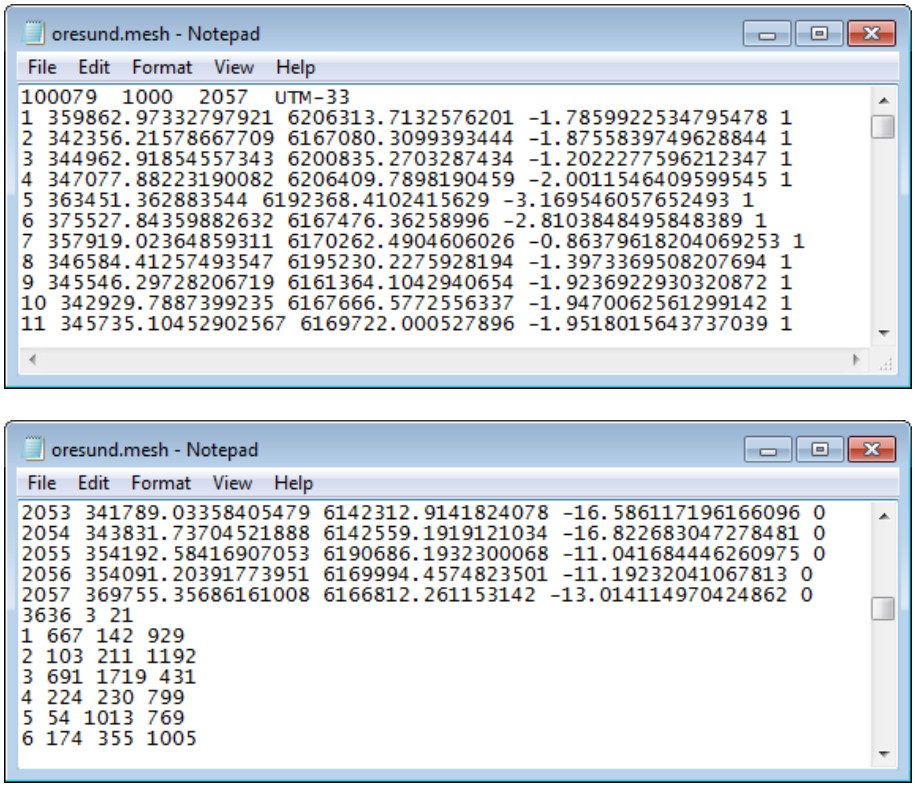


Figure 1.3 Pieces of the mesh file 'Oresund.mesh' as shown in Notepad

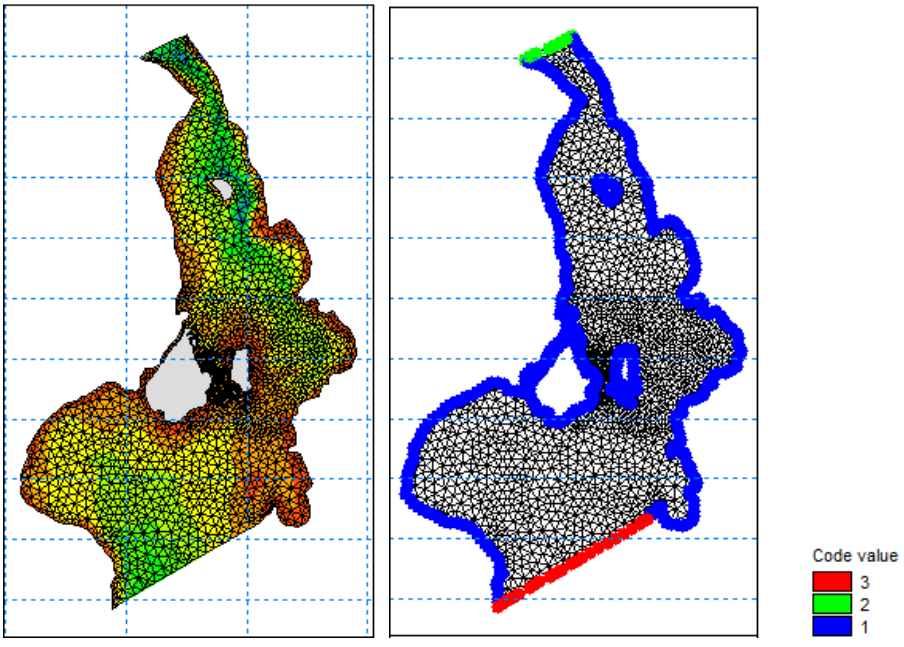


Figure 1.4 The mesh file 'Oresund.mesh' as shown in Data Viewer.
Left: bathymetry shown by shaded contour
Right: Code values (Node type = 0 is not shown)

Please note that the bathymetry values are defined in the nodes. Bathymetry values for the elements shown in the display are derived from interpolation of the values in the node.

2 Bathymetry Data

When creating a mesh to use in the simulation it is necessary to consider several things.

The data necessary for the mesh creation is the mesh boundary outline, defining the model domain, and related scatter data, defining the bathymetric values.

The mesh file couples water depths with different geographical positions and contains the following information:

- Computational grid
- Water depths
- Boundary information

Creation of the mesh file is a very important task in the modelling process.

Creation of the Computational Mesh typically requires numerous modifications of the data set, so the processes is a repetitive process that cycles until an optimal mesh resolution has been obtained.

2.1 Computational Grid

The computational grid defines the domain area and the contained mesh resolution influence the accuracy and duration of the numerical simulation.

The extent of the domain area must be decided based on the purpose of the simulation; the area of interest should be so far away from the open boundaries that any modifications in the area of interest will not affect the boundaries.

2.2 Water Depths

The bathymetry is usually interpolated from xyz scatter data holding a water depth value. These are typically found from measurements, surveys, digitisation of maps or exported from MIKE C-MAP.

In this example the applied scatter data is as follows:

- Regional lines from 1996 (measurements KI)
- Local lines 1996 (measurements KI)
- Bathymetry survey at harbour entrance (1996)
- Water points from MIKE C-MAP

2.3 Boundary Data

It is important for the further use of the mesh in numerical modelling that the boundaries of the domain are defined such that land is divided from other types of boundaries and that the boundaries contain a unique number to be identified by.

The boundary outline consists of polylines, which can be derived from manual measurements, open source data or from exports from MIKE C-MAP. In this example the pre-2000 land boundary data is digitised manually from a map, whereas the post-2000 land boundary is derived from C-MAP.

2.4 Background Images

It is possible to import DHI georeferenced images as background layers into the Mesh Generator. Using this option will enhance the understanding of the numerical model implementation in the domain as well as aid in the quality assurance of the necessary mesh resolution and individual node positions.

A number of DHI georeferenced images have been supplied for this example. The pre-2000 images display geographical maps or survey information, whereas the post-2000 images are derived from Google Earth.

3 Initial Computational Mesh

In order to get started, the first task is to create a simple mesh that will show the outline of the necessary area and form the basis for further refinement.

For this task the main focus is on the pre-2000 conditions.

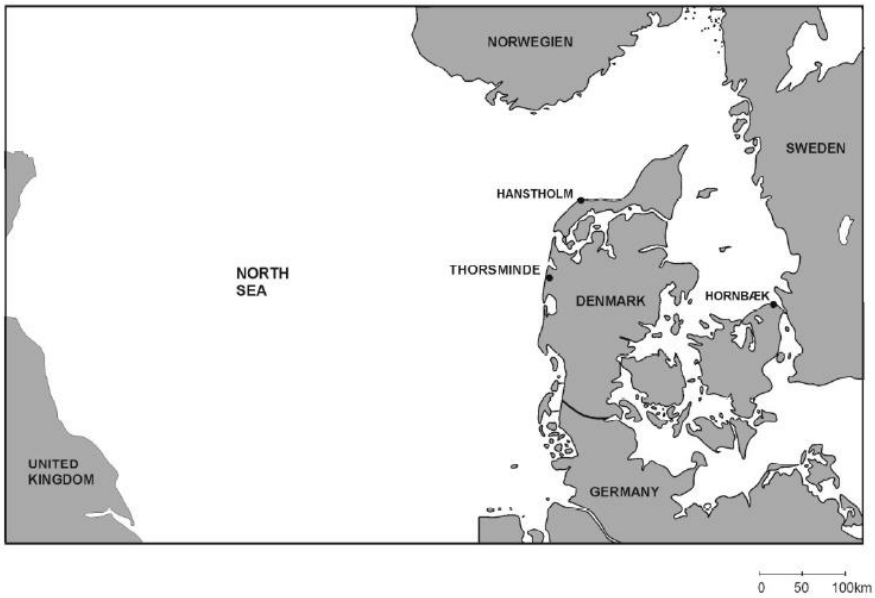


Figure 3.1 Location of the area of interest: Thorsminde, West coast of Denmark

3.1 Creating the Workspace

The mesh file containing information about water depths and mesh is created with the Mesh Generator tool in MIKE Zero. First you should start the Mesh Generator (File→New→File→Mesh Generator). See Figure 3.2.

After starting the Mesh Generator you should specify the projection system as UTM and the zone as 32 for the working area. See Figure 3.3.

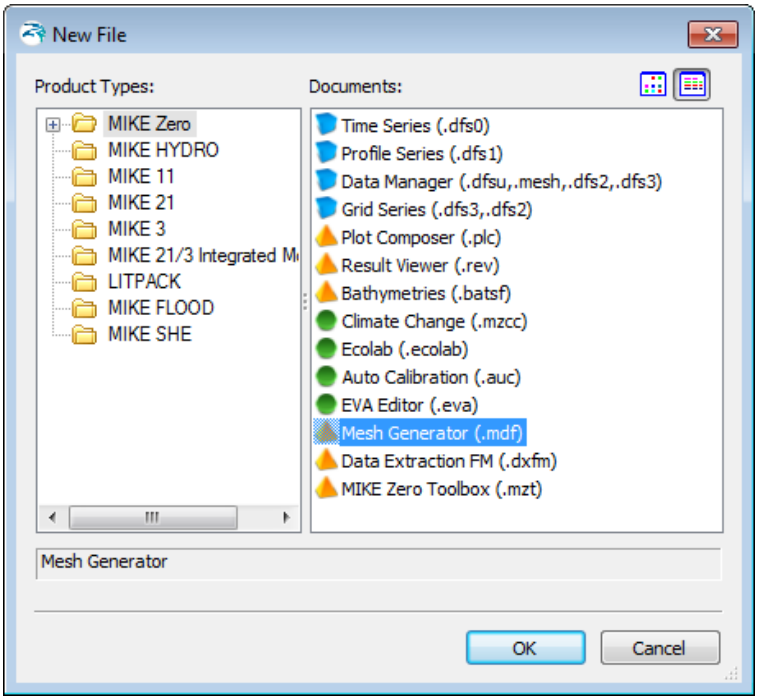


Figure 3.2 Starting the Mesh Generator tool in MIKE Zero

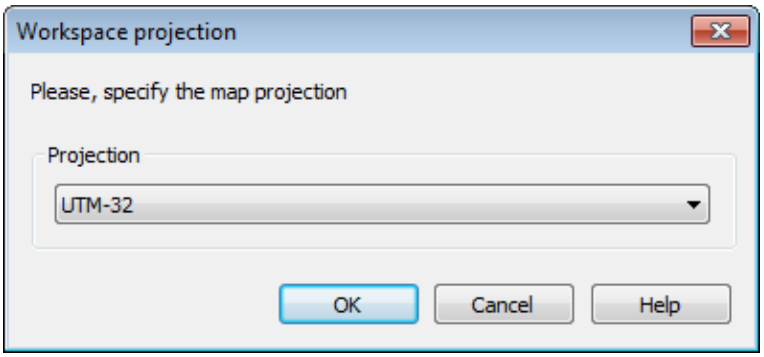


Figure 3.3 Defining the projection of the workspace

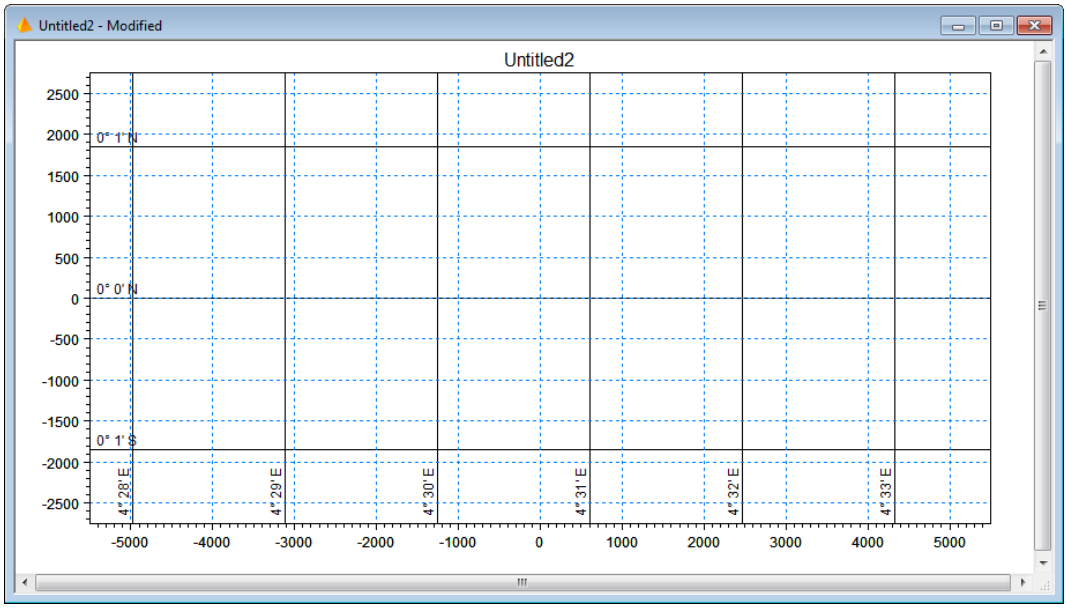


Figure 3.4 The workspace as it appears in the Mesh Generator after choosing projection

In order to visualise the area a graphic layer is imported (Options→Import Graphic Layers). Create a new layer and choose the rectified image RJylland.jpg, see Figure 3.5.

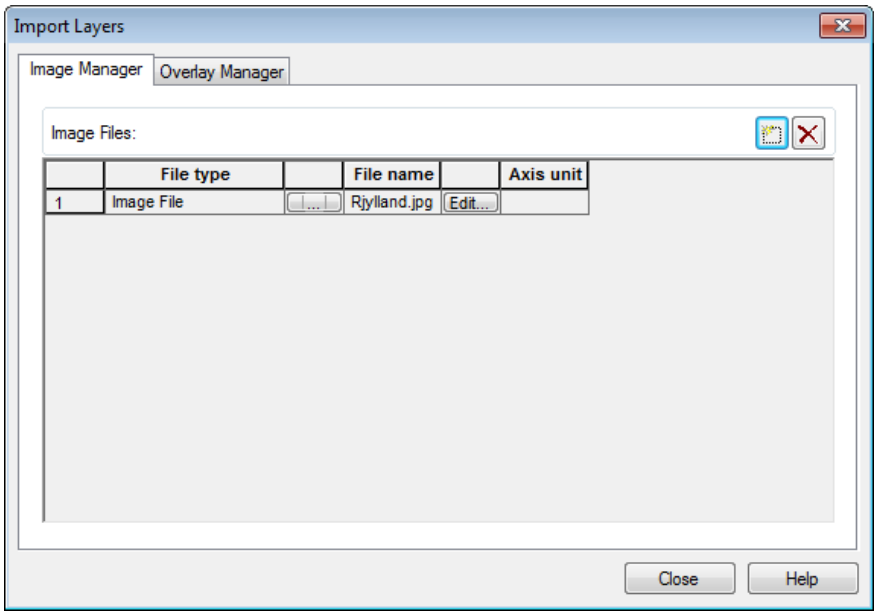


Figure 3.5 Selecting background image

At this stage you should use the Overlay Manager to change the drawing order so the image is drawn first (Figure 3.6).

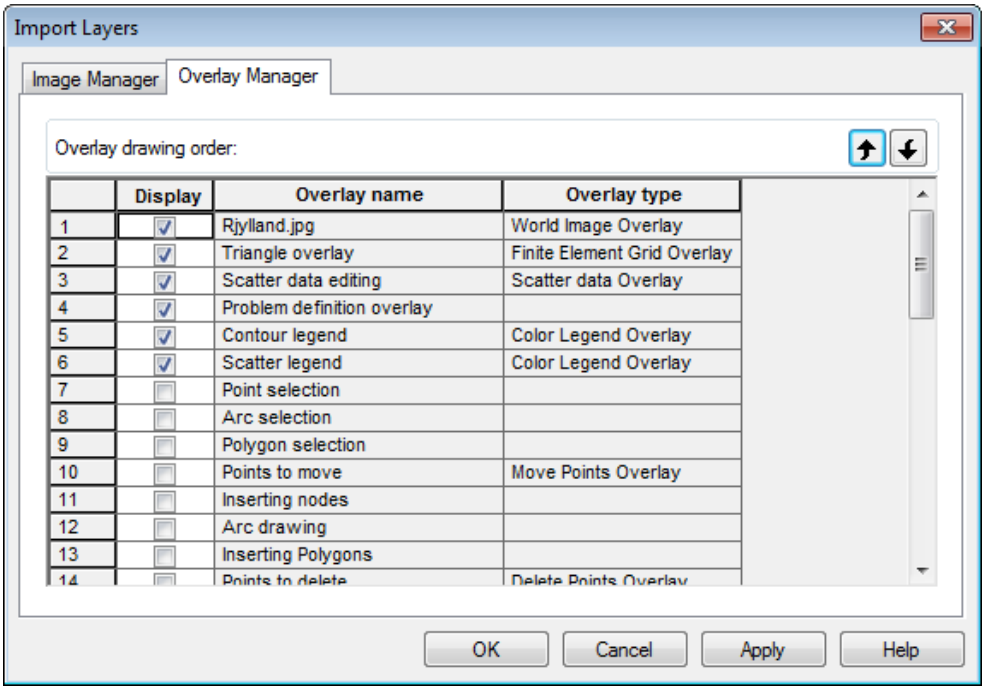


Figure 3.6 Defining imported background image at the top in Overlay Manager

Please note that you cannot see the image in the display until you define a workspace that entail it (Options→Workspace…).

Define the workspace area as shown in Figure 3.7. This should cover the area of interest.

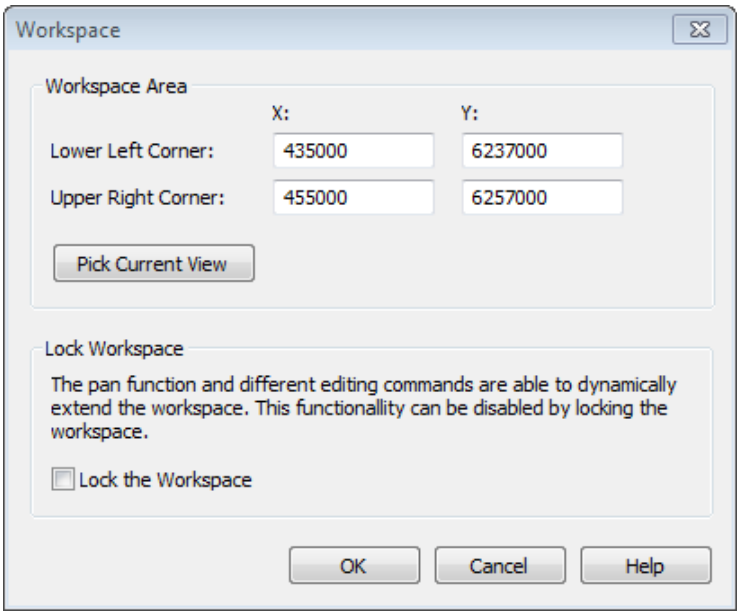


Figure 3.7 Defining workspace area

Select ‘Zoom out’ in the toolbar to visualise the entire workspace (see Figure 3.8).

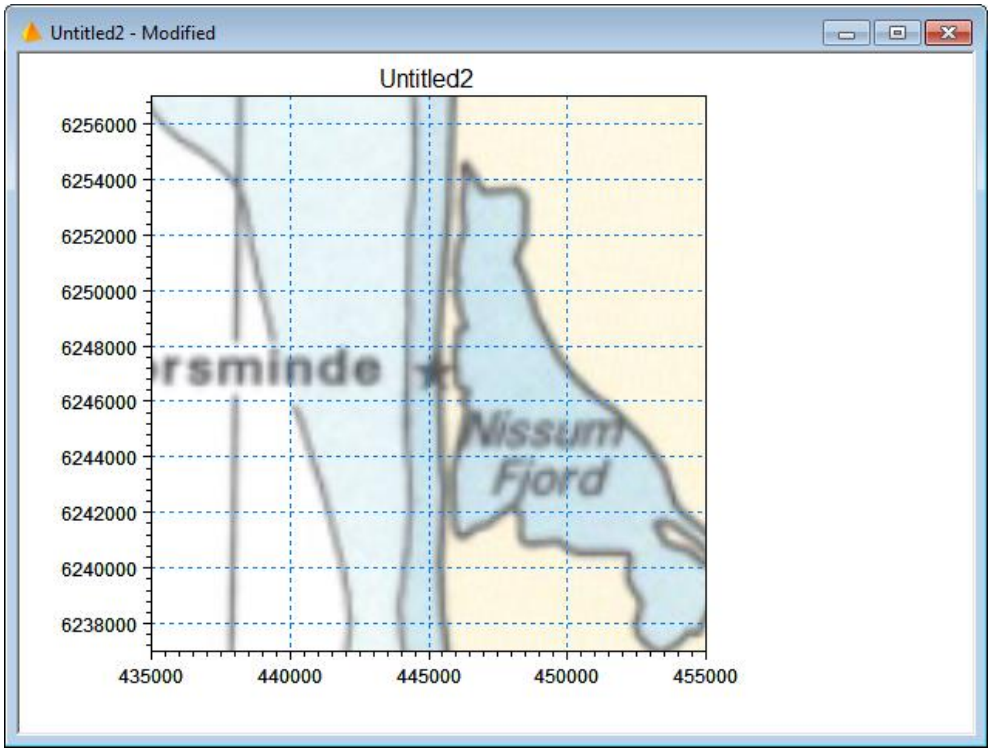


Figure 3.8 The workspace as it appears in the Mesh Generator after importing image and modifying workspace

In order to evaluate the outline of the model area import all the available scatter data (Data→Manage Scatter Data →Add…). Note that the map projection for the individual xyz files is defined during import. Press 'Apply' to refresh the display.

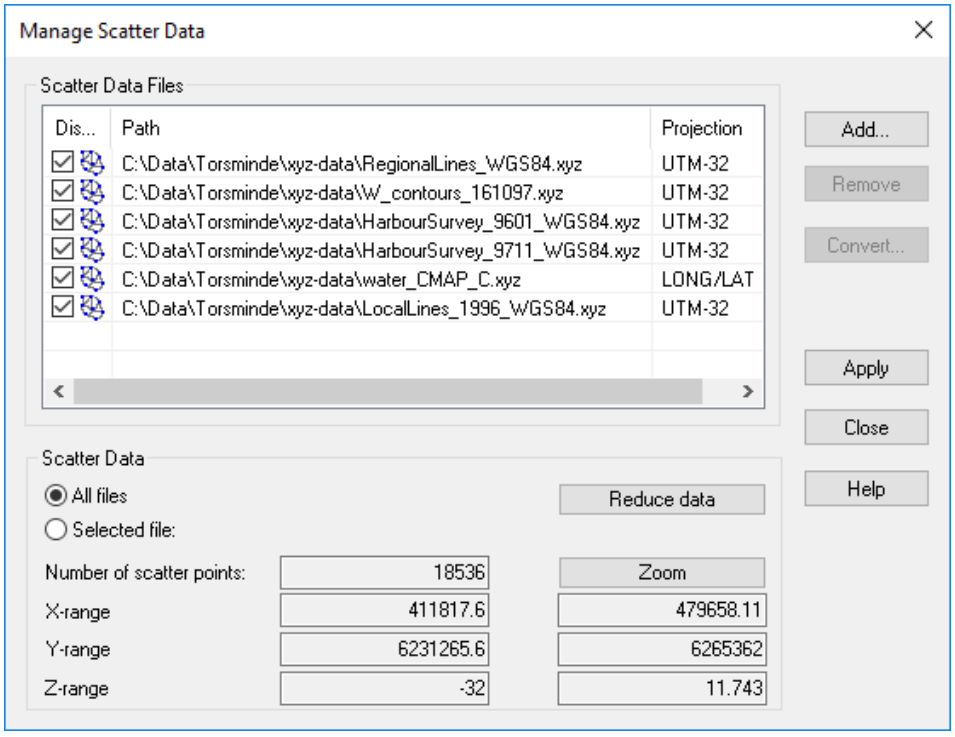


Figure 3.9 Importing scatter data. Note the projection of the individual files

The mesh is to be used for calculations in the nearshore area of the harbour. The water depths in the nearshore area are well defined by surveys while in the regional area the bathymetry is defined by navigational charts. You may decide to reduce the workspace area at this point.

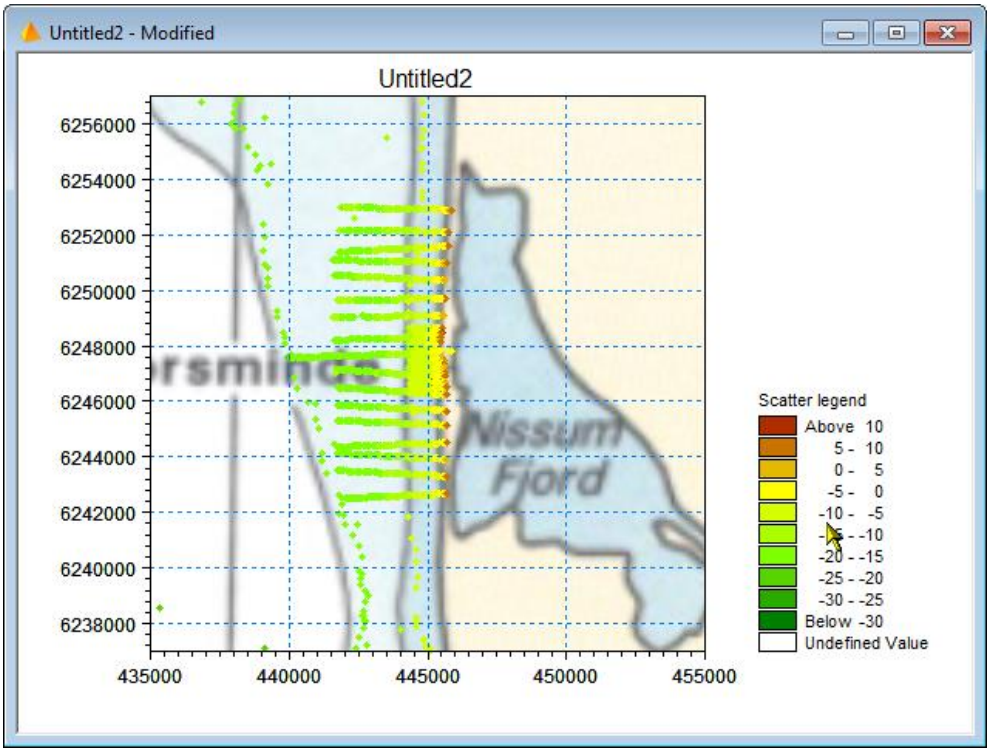


Figure 3.10 The workspace as it appears in the Mesh Generator after importing scatter data

3.2 Digitise Model Boundary

Include the image file RThorsminde_OldHarbour.jpg in the display. This file contains a more detailed image of the harbour area. The background images is now used as basis to digitise the land contours.

Select the 'Draw arc' functionality and draw the outline of land contours. This could consist of a number of arcs.

The scatter data values can be used to evaluate the position of the land contour in areas where the background image is less accurate. Note that the position of the land contour is outlining the simulation domain only; it is not defining the position of the MSL contour.

Figure 3.15 shows an example of the digitised contour.

After conclusion you can export the shoreline data to an ASCII file. Choose the 'Select Arc' functionality, right-click the workspace, select 'Inside rectangle' in the pop-up dialogue and outline the area from which you want to export the shoreline data. Export the data to an ASCII file (Data→Export Boundary →Selected Arcs→Land/Water file: e.g. myland.xyz).

You can now continue to work with your digitised land contours or import an existing boundary file following the guidelines in Section 3.3.

3.3 Import Model Boundaries

Import digitised shoreline data from an ASCII file (Data→Import Boundary →Open XYZ file: DigitizedLand.xyz). See Figure 3.11.

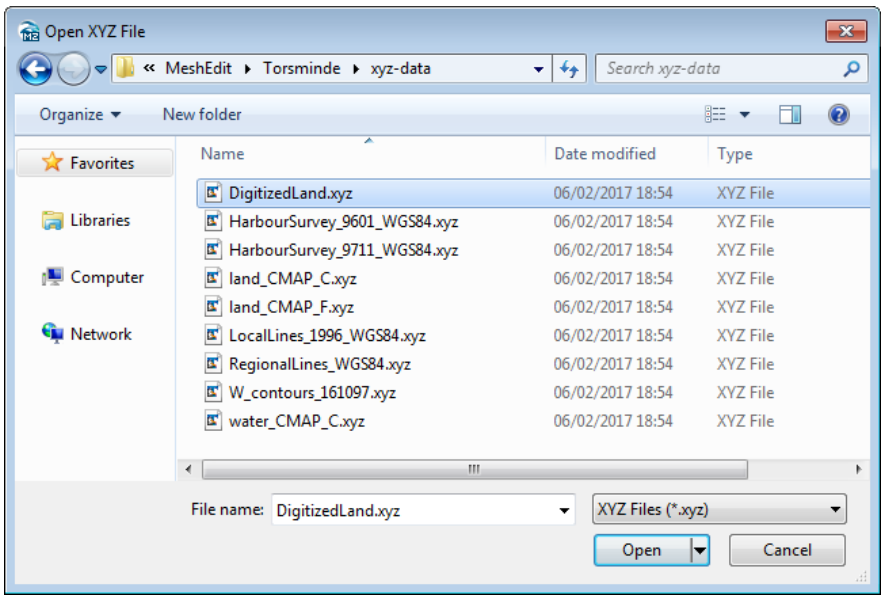


Figure 3.11 Import digitised shoreline data file

This file has been digitised using UTM-32 map projection, so this Projection should be selected. See Figure 3.12.

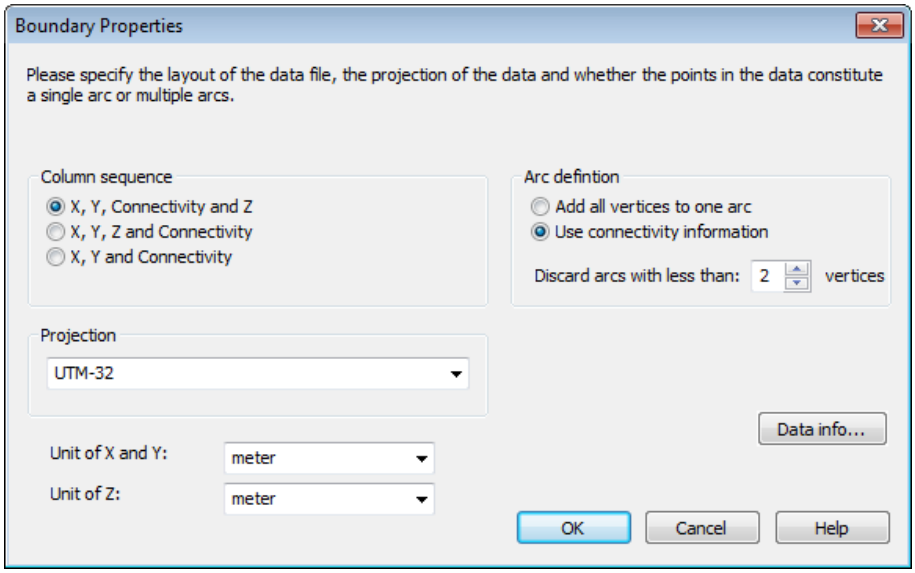


Figure 3.12 Define imported format of digitised shoreline data

The resulting workspace with the imported shoreline data will appear in a so-called Mesh Definition File (.mdf file) as shown in Figure 3.13.

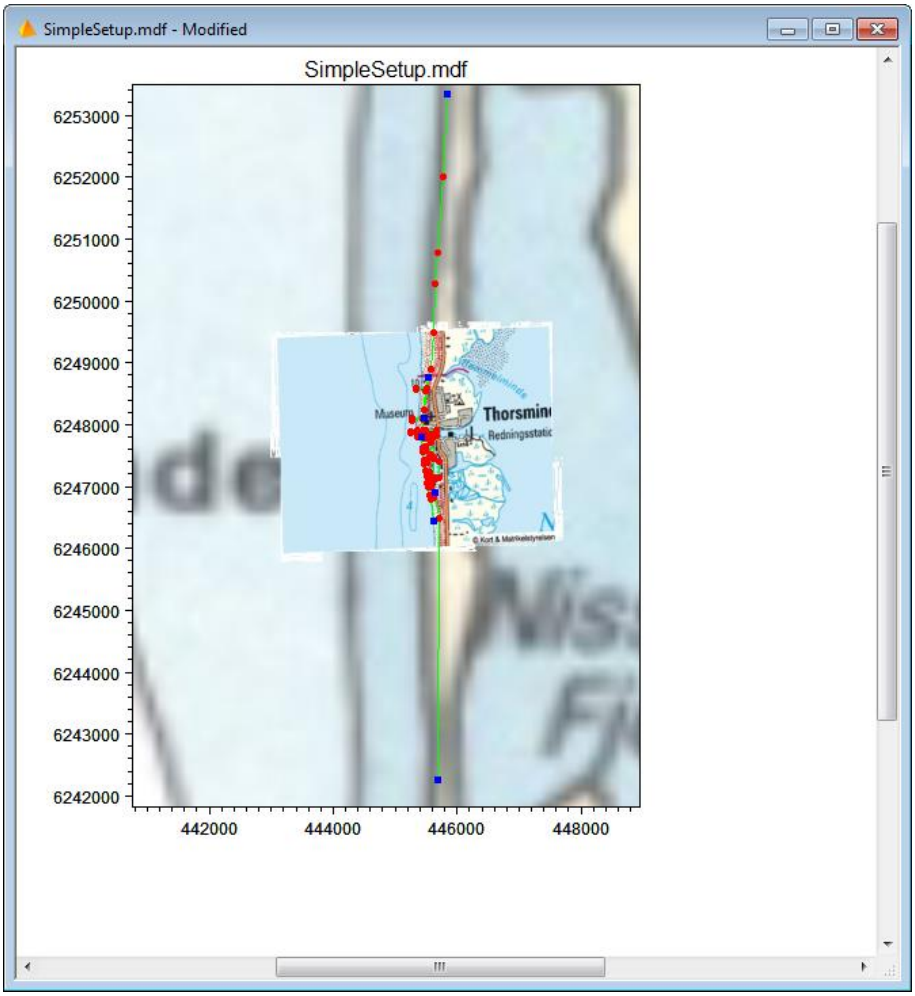


Figure 3.13 The .mdf file as it appears in the Mesh Generator after importing digitised shoreline xyz data and zooming to area

3.4 Specification of Domain

This task should result in a file with land and water boundaries forming a closed domain that can be triangulated.

3.4.1 Editing land boundary

A boundary line must be clearly defined. An imported boundary line file may contain data that is not necessary, duplicate information or crossing arcs. Thus it is always necessary to evaluate the land boundary before proceeding with the specification of the domain.

In the present case the land boundary arcs should be merged in order to reflect a closed boundary to the east. As the conditions inside the harbour is of no significance, this area should be omitted.

Start by deleting the shoreline vertices and nodes (the red and blue points) that are not part of the shoreline of the area that you want to include in the bathymetry. This includes the nodes on land that you see and the area defining the inner harbour.

For this task you use the Mesh Boundary Definition toolbar:

Zoom to the nodes (the blue squares) on the arcs and ensure that the boundary line is uniquely defined. If not, change and connect the arcs in order to achieve it.

An example is shown in Figure 3.14.

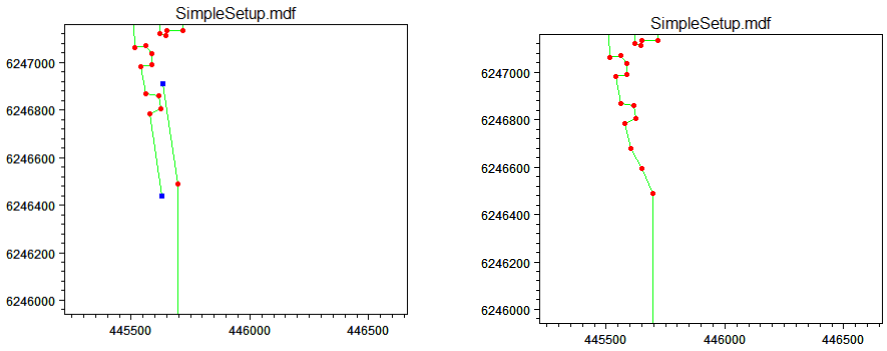


Figure 3.14 Modifying land boundary by connecting arcs and modifying points using the mesh editing toolbar

A resulting land boundary could look as follows:

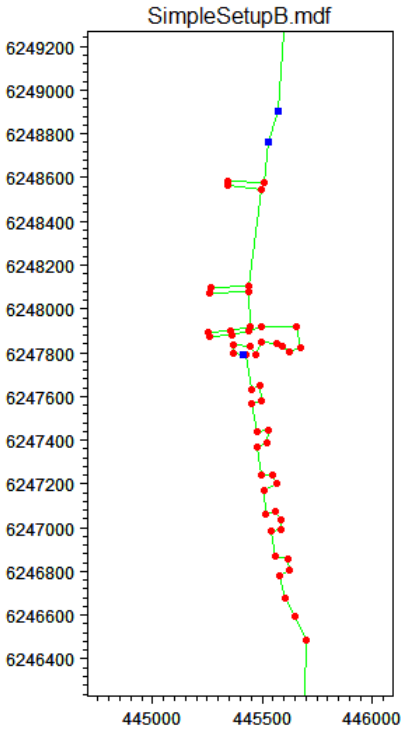


Figure 3.15 Resulting land boundary in the vicinity of the harbour

3.4.2 Adjusting the boundary data into a domain that can be triangulated

The domain area for the simulation is limited by the land boundary and three open boundaries: a north, a west and a south boundary.

The domain area is selected to extend 10 km in the north-south direction and 4 km in the east-west direction. This area is sufficiently large to avoid any influence of the boundary settings in the vicinity of the harbour. Furthermore we do have bathymetry information available to generate a good map.

Insert two nodes at the positions (442000, 6243000) and (442000, 6253000), respectively. Note, that you can modify the location of a node afterwards by selecting the node in the display, right-click → Properties…

Connect the node points by drawing arcs to close the area. These new arcs define the open boundaries. For aesthetic reasons you can move the land nodes on the north and south boundary so the domain becomes nearly rectangular. This is however not necessary.

The resulting domain may look as shown in Figure 3.16.

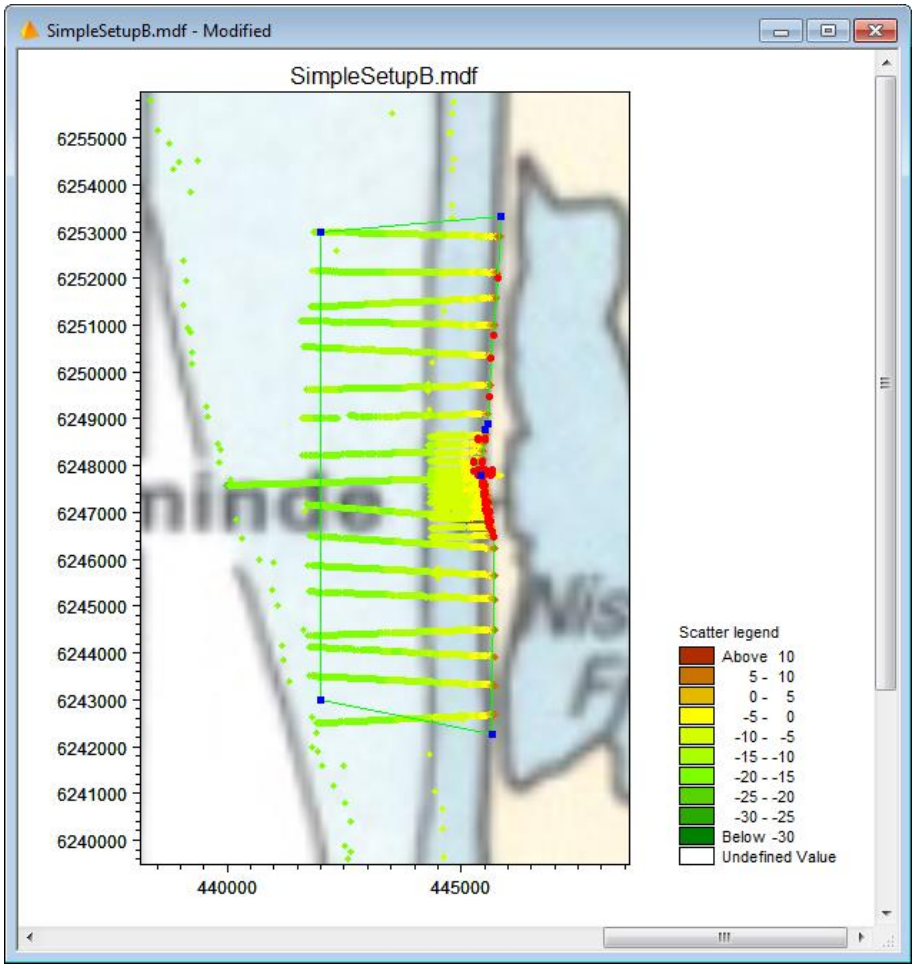


Figure 3.16 Defined domain area shown with background image and position of scatter data

3.5 Specification of Boundaries

The node points and arcs on the open boundaries must be defined by a unique integer value. These attributes are used for the model system to distinguish between the different boundary types in the mesh: attributes equal to 2 and above correspond to open boundaries, attribute equal to 1 correspond to a land/water boundary. Note that the attribute for a boundary will automatically be set to 1 during mesh generation if it is not already defined by a number larger than 1.

Mark the North Boundary arc, right-click and choose Properties. Set the arc and node attributes to 2 for the northern boundary, see Figure 3.17 and Figure 3.18. Mark 3 for the western boundary arc and mark 4 for the southern boundary arc, respectively. Note that the property values of the arcs are unique.

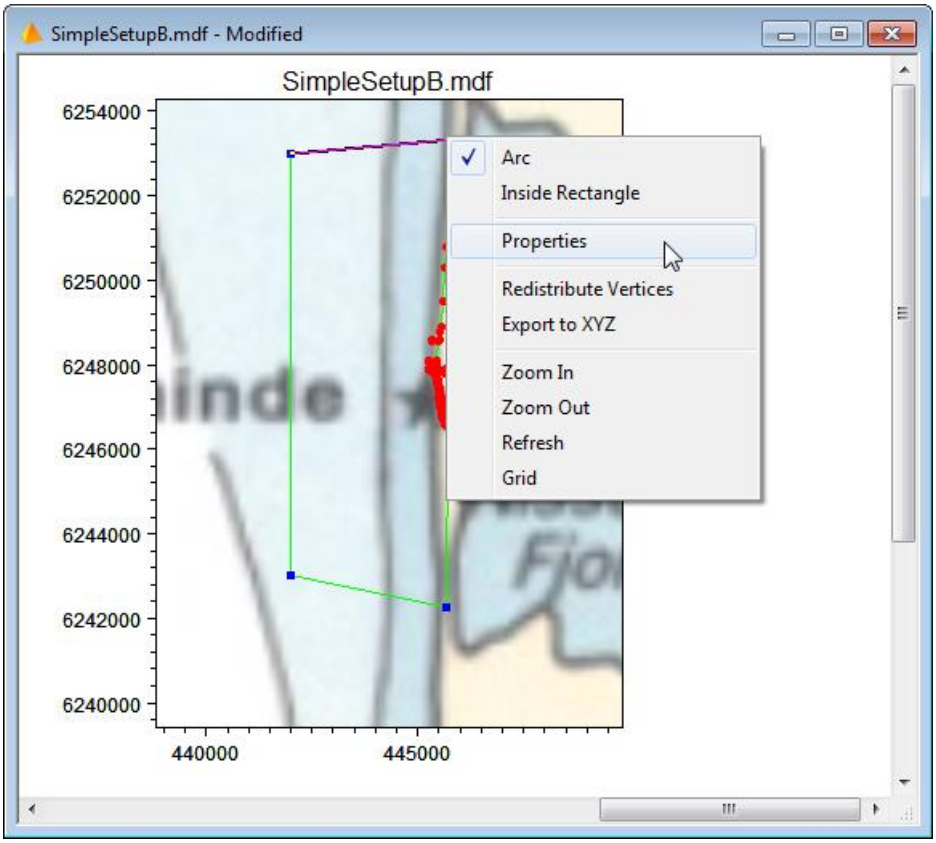


Figure 3.17 Defined domain area shown with background image. Selecting the arc for the northern boundary (purple arc) for editing properties (right click with mouse)

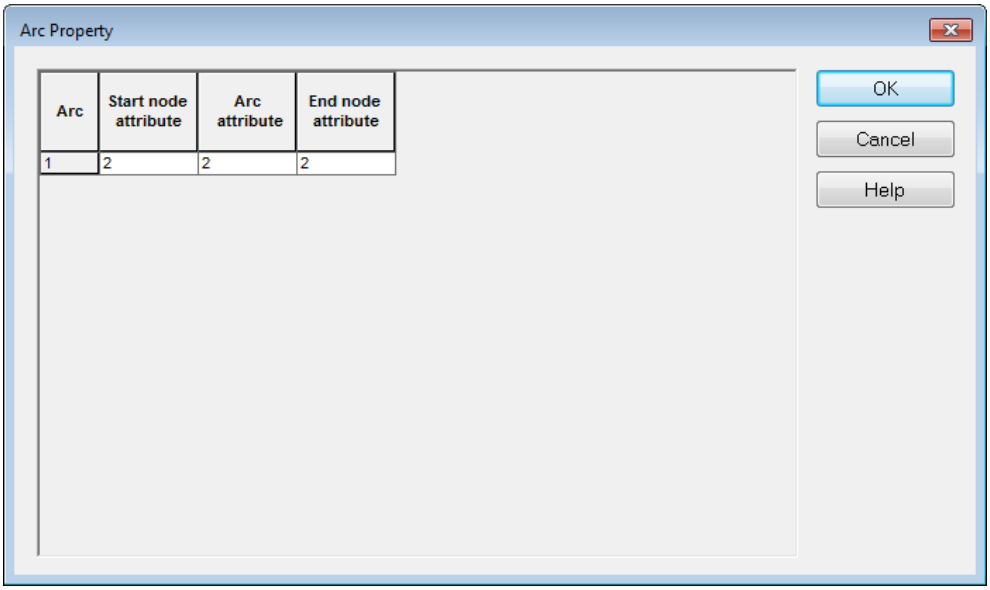


Figure 3.18 Editing the northern boundary arc properties

3.6 Mesh Generation

The next step is to triangulate the domain.

Try to make the first triangulation (Mesh→Generate mesh…). Use the triangulation option settings as shown in Figure 3.19.

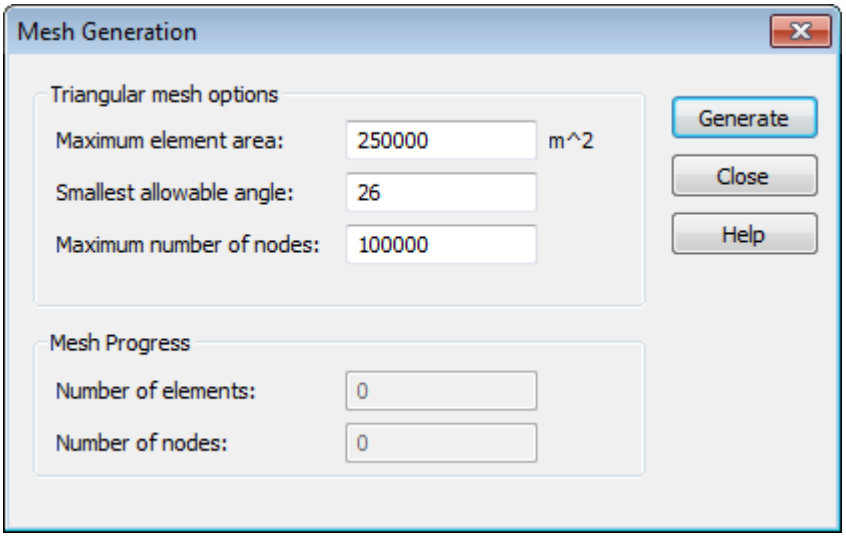


Figure 3.19 The triangulation options for a first attempt

This setting will result in fewer than 500 elements. Elements limited by boundary point positions tend to be much smaller than elements only limited by maximum element area. The resulting mesh is shown in Figure 3.20.

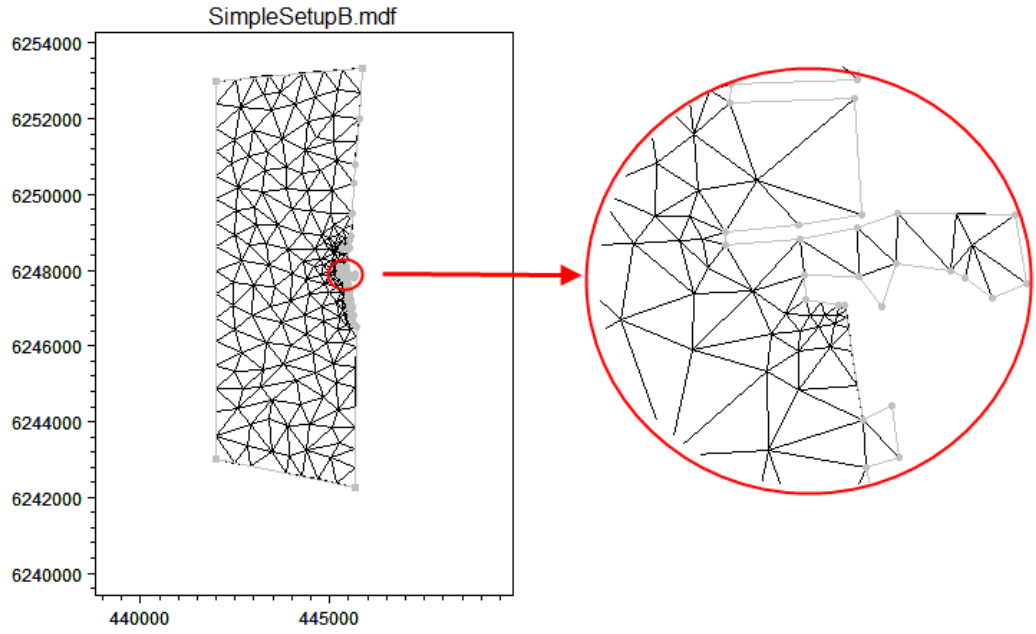


Figure 3.20 Triangulated mesh for a first attempt

After the triangulation you can use a tool for smoothing the mesh (Mesh→Smooth Mesh). In this example the mesh has been smoothed 100 times. The resulting mesh will look as in Figure 3.21.

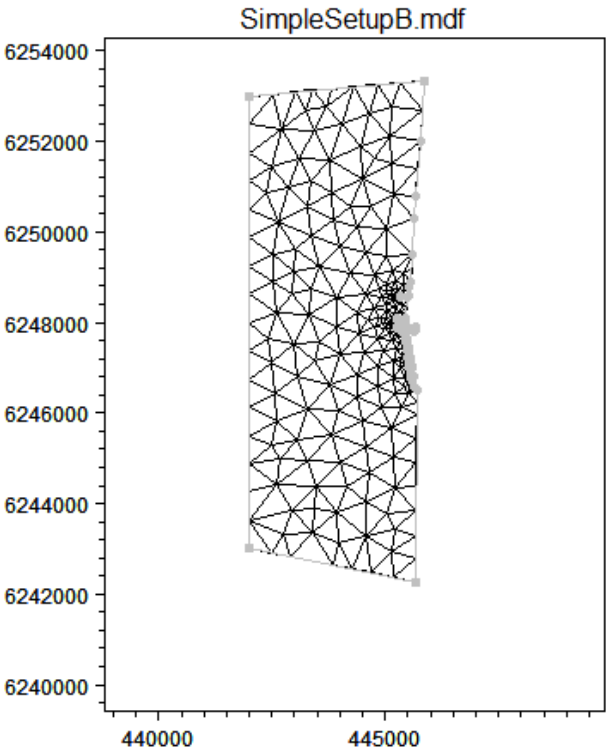


Figure 3.21 Resulting mesh after smoothing 100 times

We will consider how to refine the mesh in specific areas later in this step-by-step guide.

3.7 Analysis

After mesh generation and smoothing of the mesh, it is feasible to analyse the mesh in order to see if it can be improved. For this use the Analyse tool in the toolbar (Mesh→Analyse mesh…).

Please note that you have to be in the 'Mesh Editing' mode in order to activate the tool.

The simulation time for a setup increase with increasing number of elements, decreasing element size and decreasing time step. The angles within the elements should be as large as possible in order to optimise the solution. The analysis tool can be applied to find individual elements that are limiting the optimal solution setup.

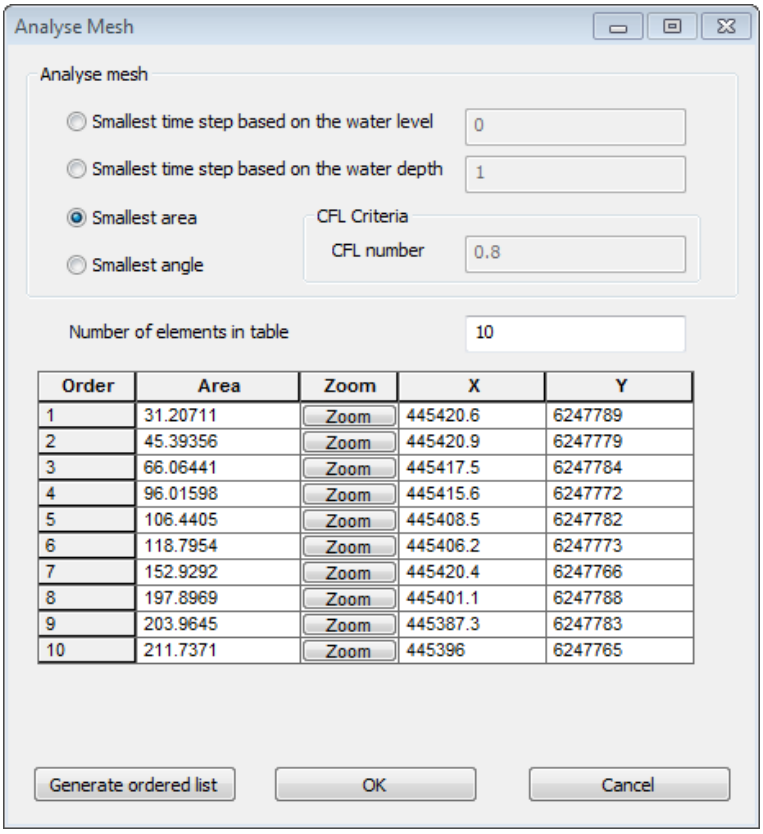


Figure 3.22 Analyse mesh

The smaller the element, the smaller the time step must be to uphold the CFL criteria. Thus isolated small elements should be avoided. In this case the land boundary has been defined using two vertices positioned near each other. By removing one of the vertices and regenerating the mesh, the smallest area value will increase and so the time step required, see Figure 3.23.

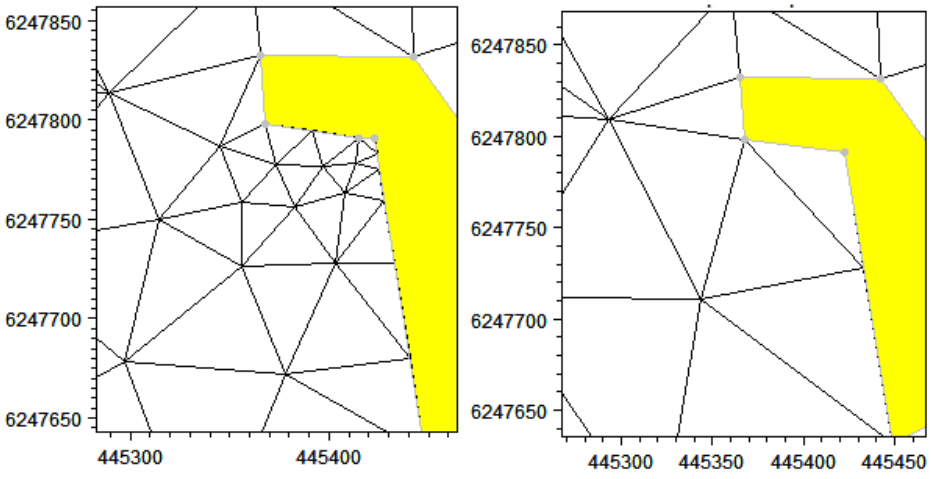


Figure 3.23 Generated mesh after removing one vertex. The smallest element is now 7 times larger

3.8 Smoothing the Mesh

In general, the mesh should be smoothed a number of times after the generation and before interpolating the bathymetry. The smoothing process will change the position of the generated node points in order to obtain the best overall resolution of the triangular elements while trying to avoid small angles in too many elements.

In the examples shown in Figure 3.24 and Figure 3.25 a smoothing using 100 times will result in a smaller minimum element angle, but overall the element angles are larger. Smoothing will only take place, if the new mesh overall is better than the original.

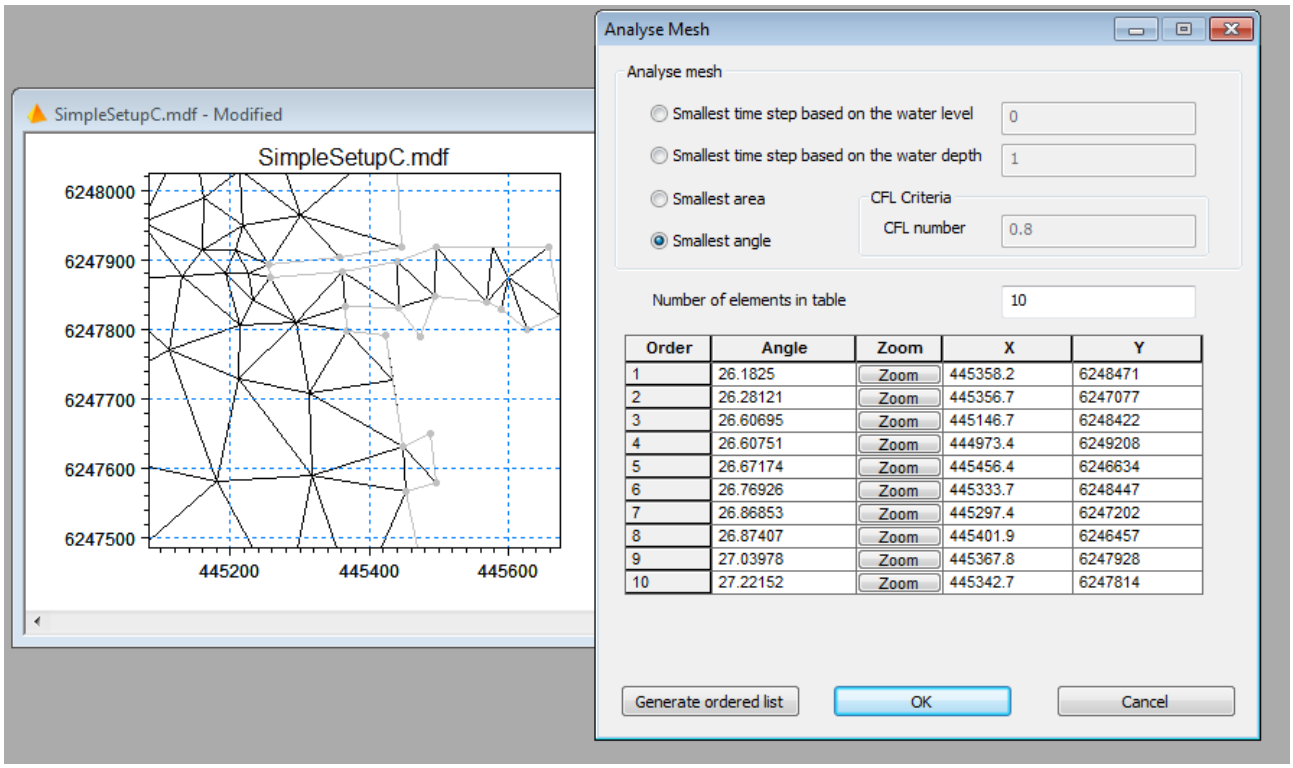


Figure 3.24 Generated mesh before smoothing. Note the 10 minimum angles found in analysis

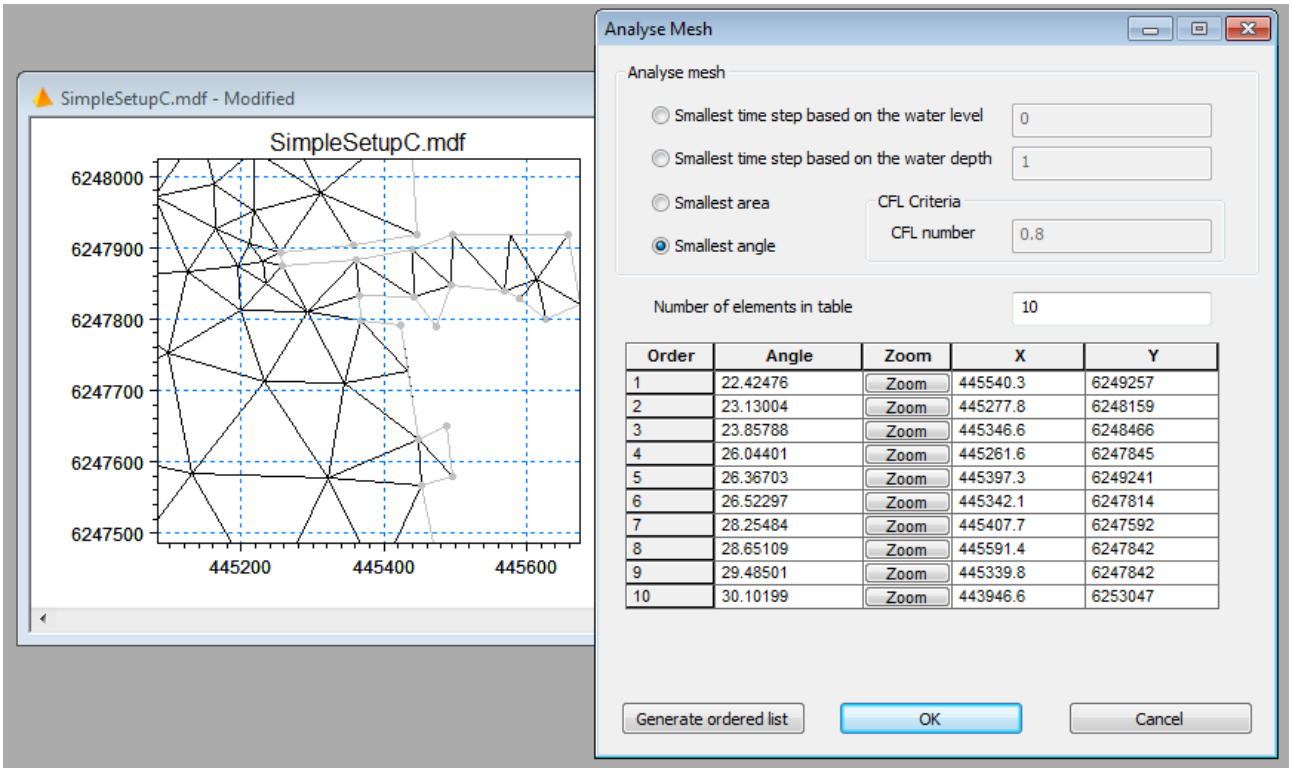


Figure 3.25 Generated mesh after smoothing 100 times

4 Initial Bathymetry

The initial bathymetry is now created from the initial mesh by interpolating scatter data. For the initial mesh only one scatter data file is applied. Thus, start with removing all previously imported scatter data files.

4.1 Importing Scatter Data

First import the xyz-file containing the water depths from the measurements of regional lines (Data→Manage Scatter Data →Add… and choose the file RegionalLines_WGS84.xyz). Specify the projection as UTM-32.

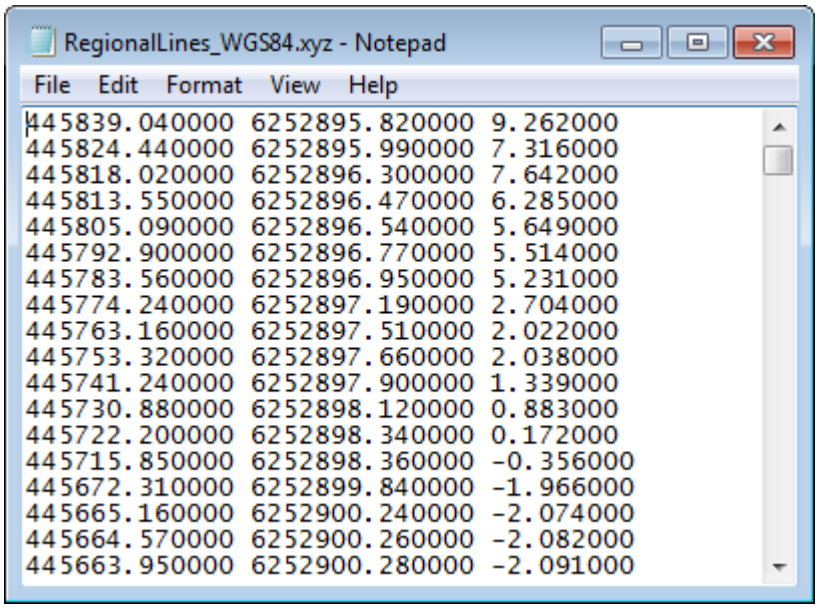


Figure 4.1 ASCII file describing the depth at specified geographical positions (Easting, Northing, Depth (and point number). Please note that if MIKE C-MAP is used you are not allowed to view the data in a text editor because the data is encrypted. If C-MAP data is used then the co-ordinates will be given in Longitudes and Latitudes

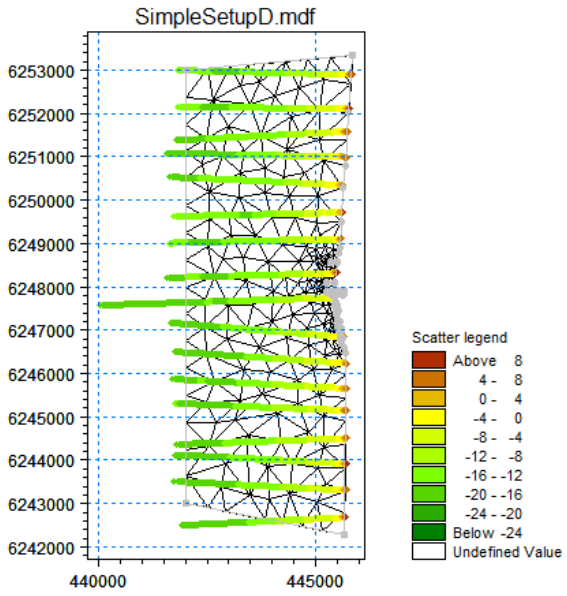


Figure 4.2 Scatter data from regional lines

4.2 Interpolating Scatter Data

Now interpolate the bathymetry values by using the default settings for the interpolation (Mesh→Interpolate…).

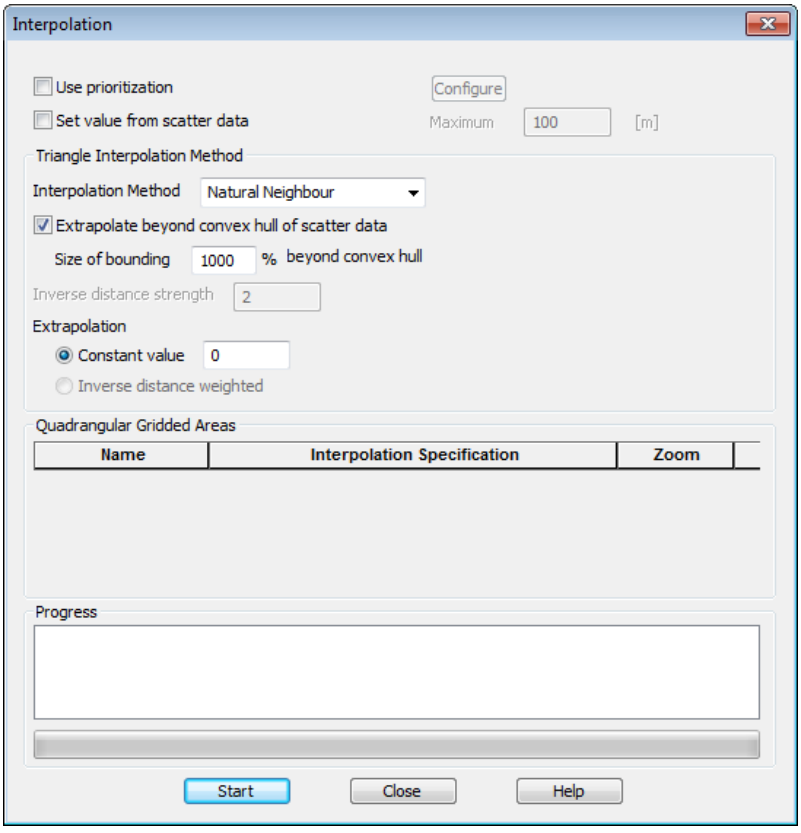


Figure 4.3 Interpolating water depths into the domain from imported XYZ file

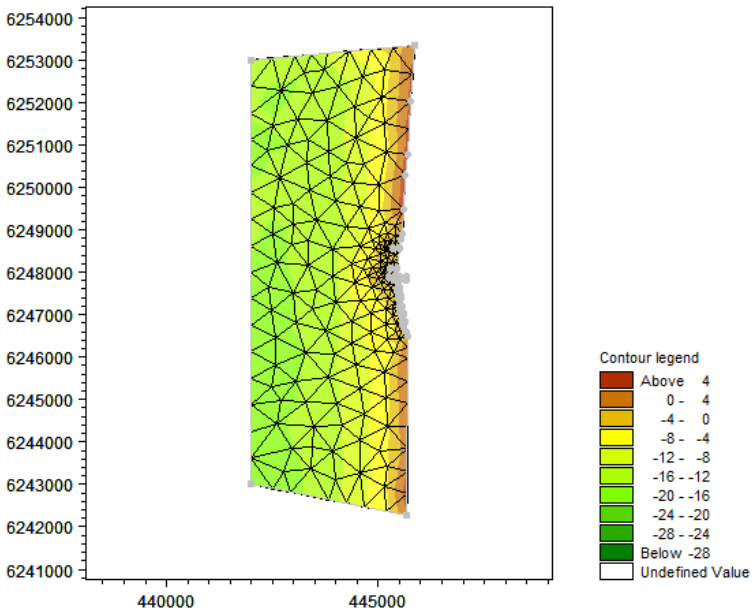


Figure 4.4 The mesh as it appears in the Mesh Generator after interpolating water depth xyz data into the mesh

4.3 Analysing the Mesh

After the interpolation of the bathymetry values, use the Mesh Analyse tool to evaluate the mesh again, now investigating the element determining the smallest time step, considering the water depth. This analysis may inspire to further improvements of the mesh.

4.4 Exporting the Mesh

Now export the mesh from the Mesh Generator (Mesh -> Export Mesh). Provide a name for the mesh file in the Export Mesh dialogue and press OK.

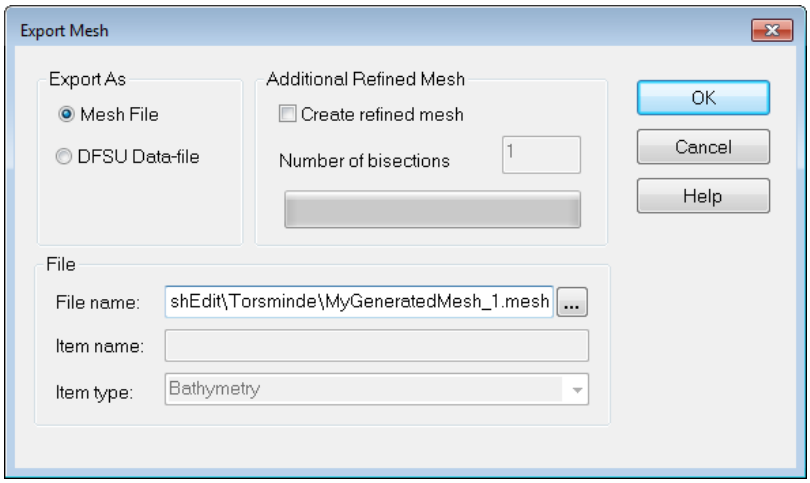


Figure 4.5 Exporting mesh to file

4.5 Viewing the Mesh

You can view (and edit) the resulting mesh file in the Data Viewer (Figure 4.7) or the Data Manager (Figure 4.8) or view it in Plot Composer or MIKE Animator Plus (Figure 4.9). MIKE Animator Plus is a very good tool to investigate the bathymetry especially for spikes, strange ripples, decreasing depth just inside the boundary etc.

The default editor/viewer for mesh/dfsu data files can be set to either the Data Viewer or the Data Manager. You use the File Association dialogue (File -> Options -> User Settings) to choose the default editor (Figure 4.6).

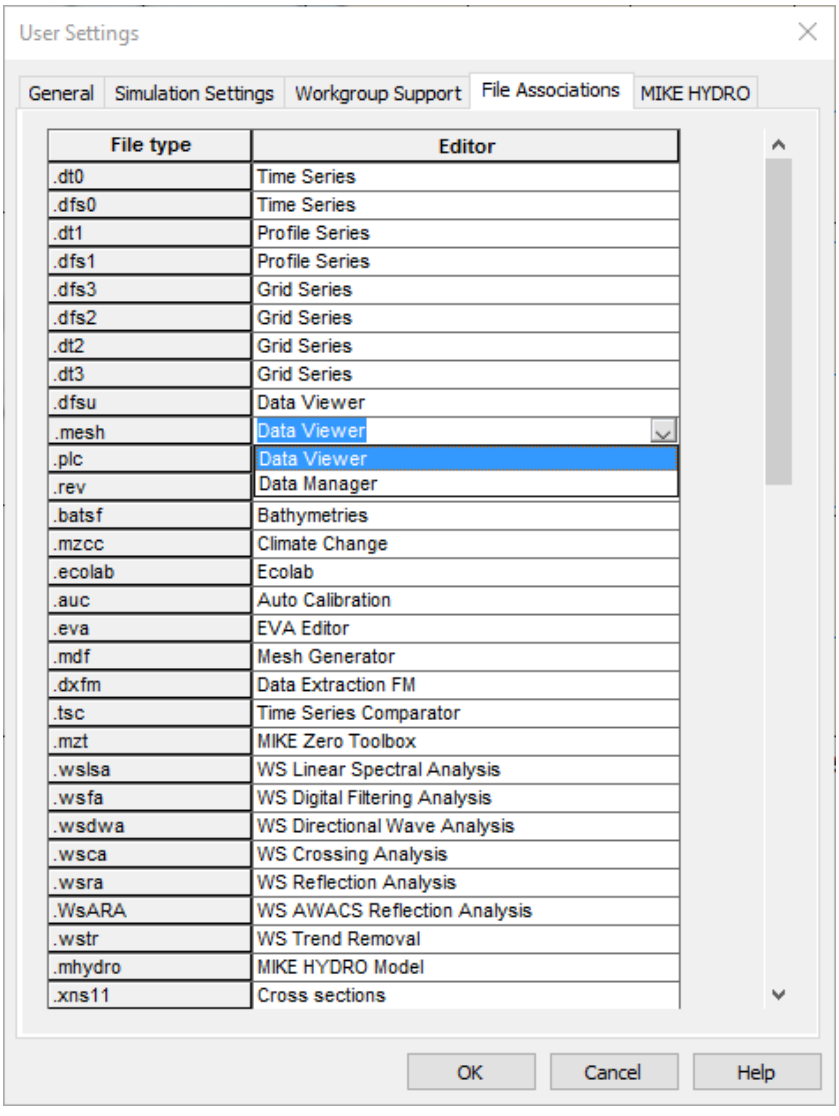


Figure 4.6 The File Association dialogue

Please note that you may open the file with the alternative editor by right-clicking the file in the Project Explorer and select ‘Open With’ instead of ‘Open’.

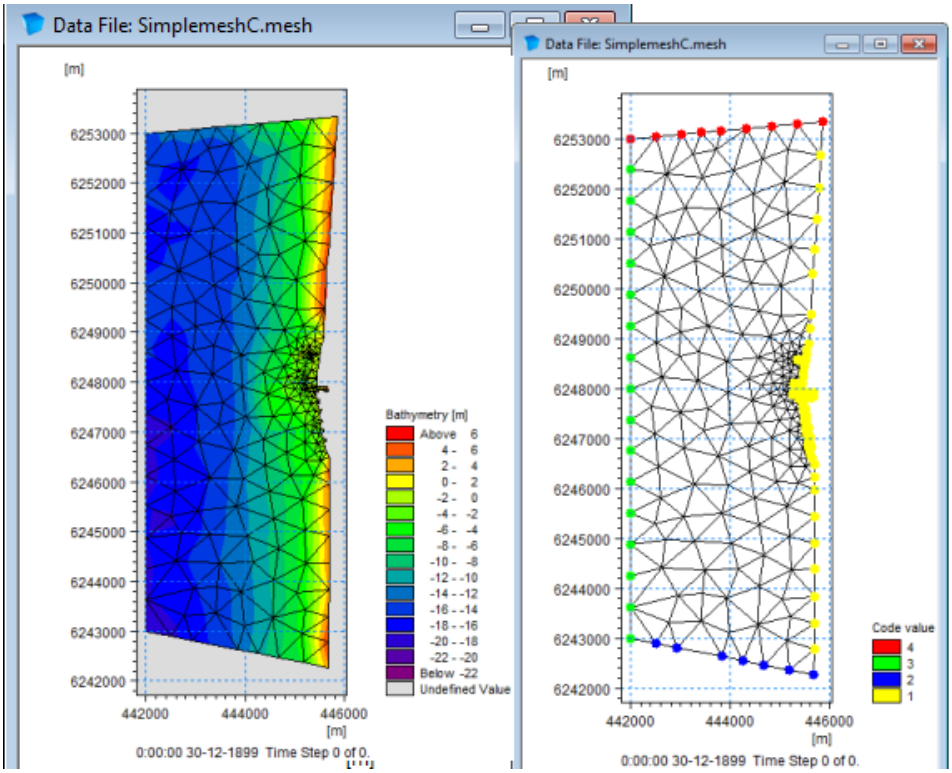


Figure 4.7 The interpolated mesh file as it appears in the Data Viewer. Left figure show the bathymetry values (shown as smoothed contour), right figure show the node type values for the boundaries

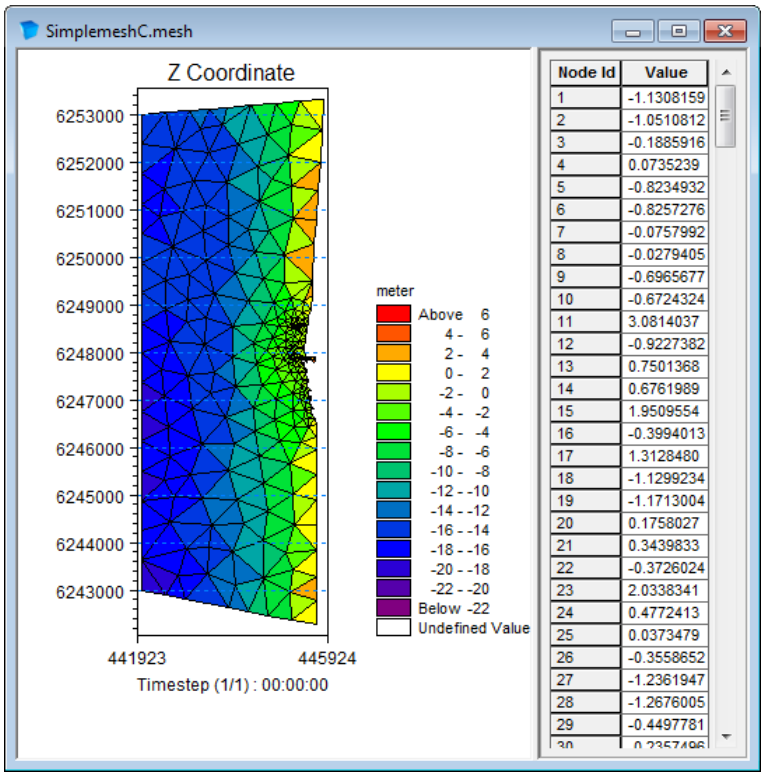


Figure 4.8 The interpolated mesh file as it appears in the Data Manager. Bathymetry shown by box contour

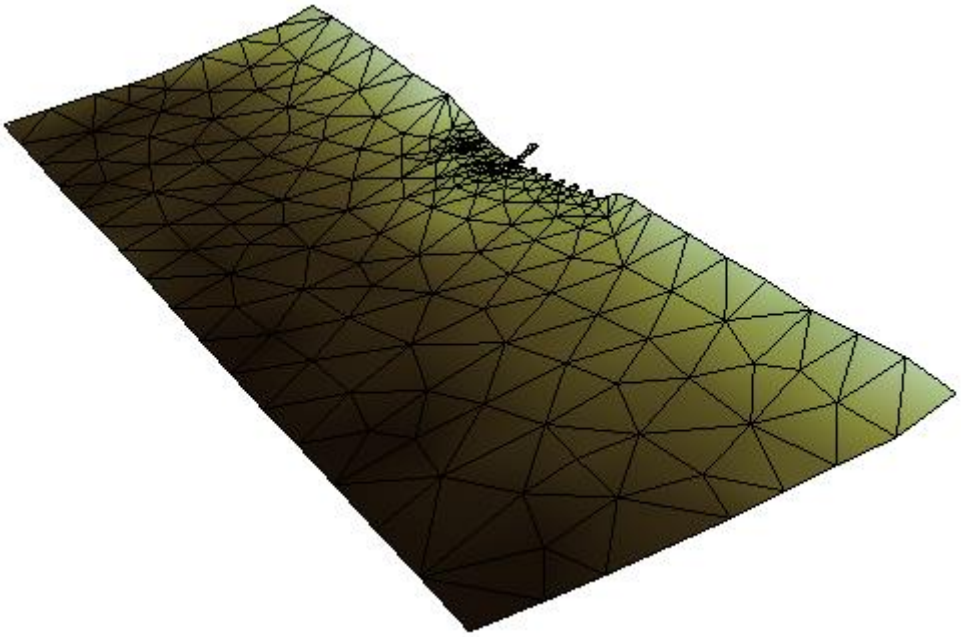


Figure 4.9 The interpolated mesh as it can be presented with MIKE Animator Plus

4.6 Next Steps

The initial domain describes an area of 4 km x 10 km, with bathymetry values from -20 m up to + 6 m. The positions of the structures near the harbour (groins and revetments) are outlined in the mesh.

The mesh generator setup for the initial resulting mesh is just a starting point for optimising the mesh to the final stage. In the following the initial mesh setup is modified in a number of ways to achieve a better representation of the bathymetry for the given simulation.

5 Mesh for Regional Wave Simulation

Suppose the aim of the simulation is to investigate the overall wave conditions and the transition of waves from deep water onto a position in front of the harbour entrance.

In this case the details of the land contour is of less importance as the land contour does not affect the wave conditions. Also, in order to evaluate the transformation of the waves in detail, the number of elements in the deep area must be increased.

5.1 Modifying the Land Boundary

The elements near the land boundary depends highly on the resolution of the boundary: for each vertex on the boundary line, a node will be generated. Thus by modifying the number of vertices on the boundary line the mesh resolution will change accordingly in the adjacent areas, cf. Figure 5.1 and Figure 5.2.

For example, if your line consist of many points, it may be feasible to re-distribute the vertices on the line using a defined number of vertices or a defined distance. This will smooth the boundary line and restrict the number of elements. Note that the Preview button gives you the opportunity to preview the outline of the new boundary before deciding to continue.

The overall wave conditions will not be affected by details near land, thus the land contour can be modified to the simplest possible. In this case the groin tip can be defined as a single node point and the coastline near the revetment can be modified to a straight line.

Delete the generated mesh to modify the land boundary. The land boundary can be edited by deleting the individual vertices on the arc.

Please note that removing the indication of the harbour would give even better results. However in this instance the harbour is kept for educational purpose.

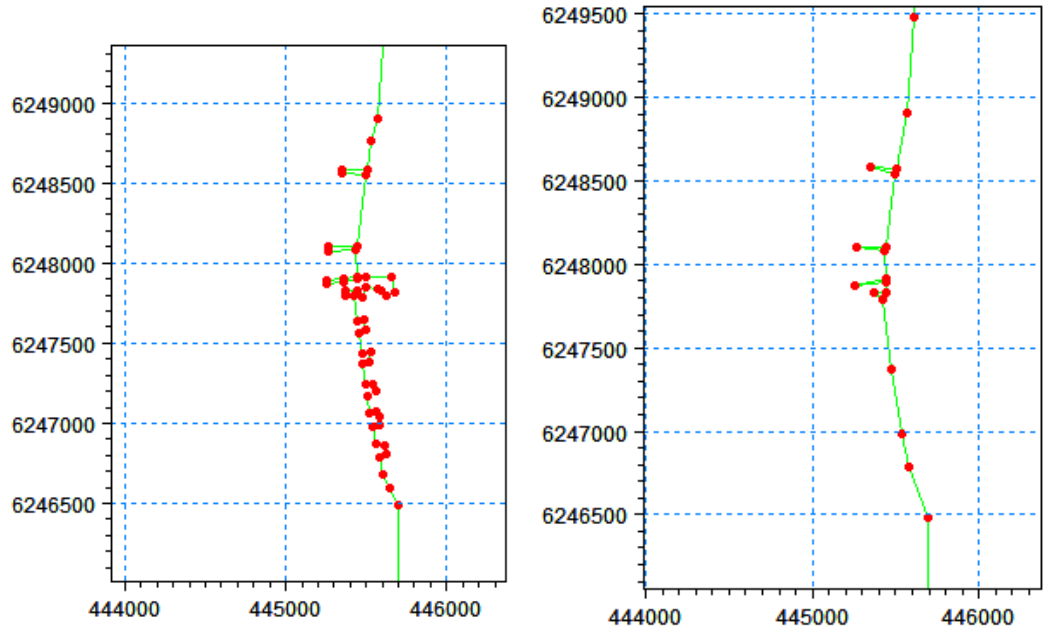


Figure 5.1 The land boundary. Left: initial boundary, right: after modifying

After modifying the land boundary, the mesh may be generated and smoothed in order to see the effect of the changed setup. An example is shown in Figure 5.2.

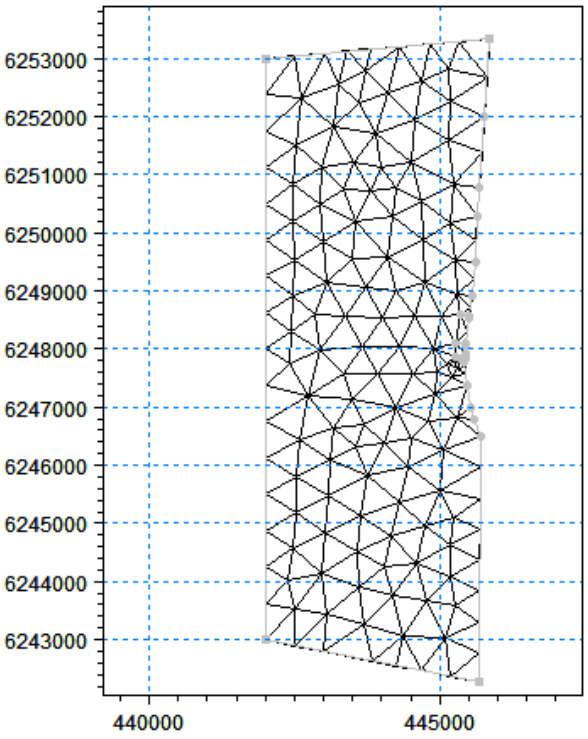


Figure 5.2 Resulting mesh after modifying the land boundary

5.2 Increasing Resolution

In order to evaluate the wave transformation in the area, the mesh resolution must be more refined.

5.2.1 Simple refinement of mesh

By refinement through bisection the existing mesh is used as basis for the refinement. A number of face bisections determines how many times each element is to be divided into 4 smaller elements, thus creating a more refined mesh. Note that for each bisection, the total number of elements in the mesh is multiplied by a factor of four.

An example of the mesh after refinement using bisection 1 time is shown in Figure 5.3.

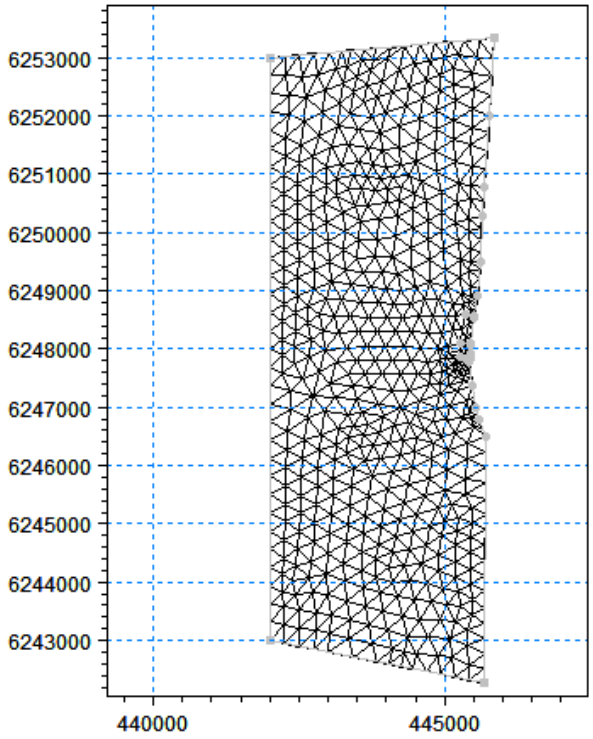


Figure 5.3 Resulting mesh after refinement using bisection one time (1092 elements)

Please note that in this way it may still be the much smaller element areas near the harbour entrance that will determine the simulation time.

5.2.2 Using smaller maximum element areas

A better way is to decrease the maximum element area used during the mesh generation. By reducing the maximum element area by a factor of four, i.e. from 250000 to 62500, the mesh resolution in the deeper areas resembles the case using bi-section but the creation of smaller areas near the harbour is avoided, thus allowing for larger time steps in the simulation.

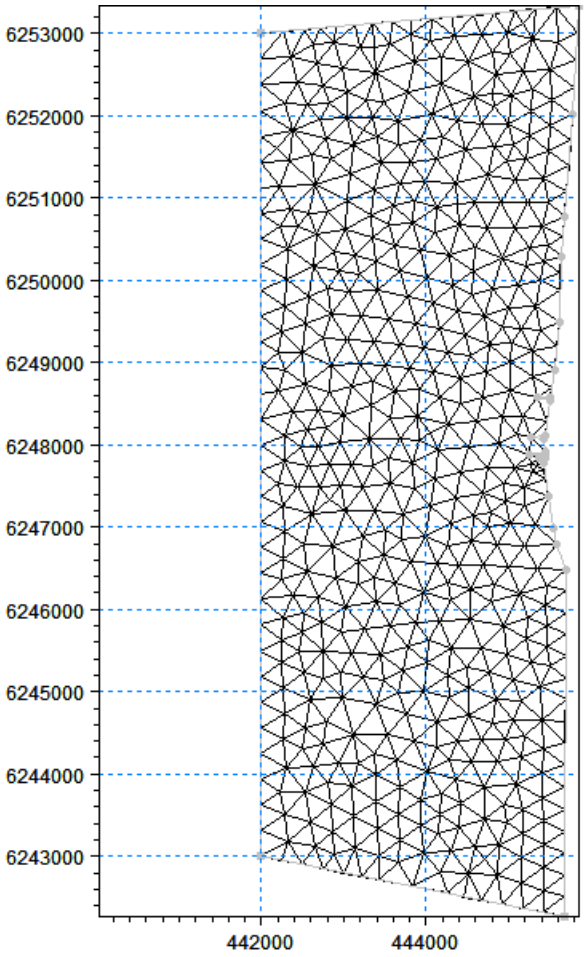


Figure 5.4 Resulting mesh after refinement using smaller maximum element area (966 elements)

5.3 Interpolating Bathymetry

The mesh resolution still seems a bit too coarse for the wave simulation, so in the following a mesh with maximum element area 30000 m^2 is applied, corresponding to 2026 elements.

In the present case the focus is on the simulation of overall wave conditions. For this the bathymetry information from the regional lines will contain the information needed.

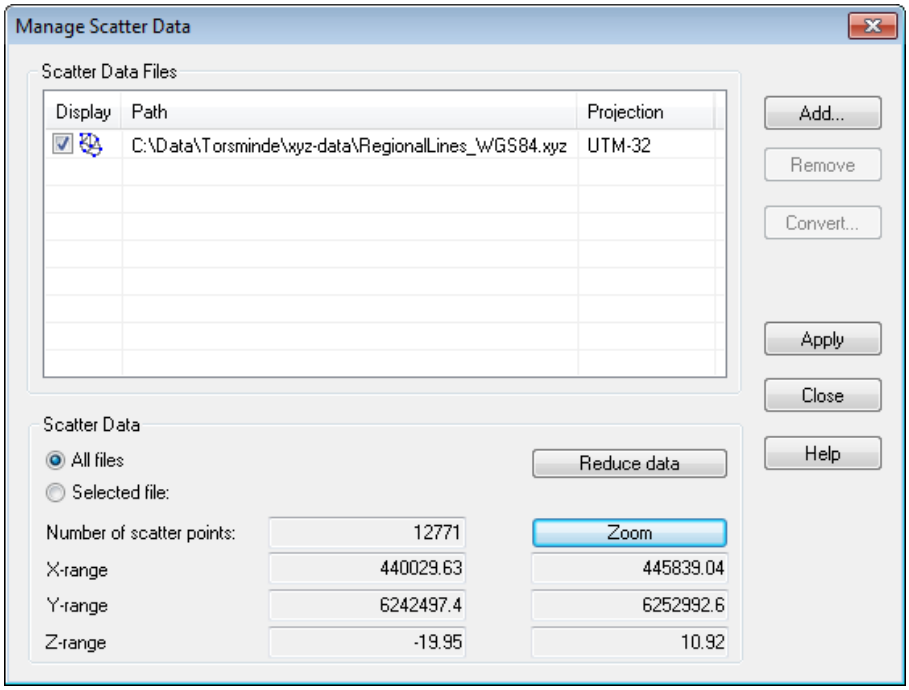


Figure 5.5 Importing of scatter data.

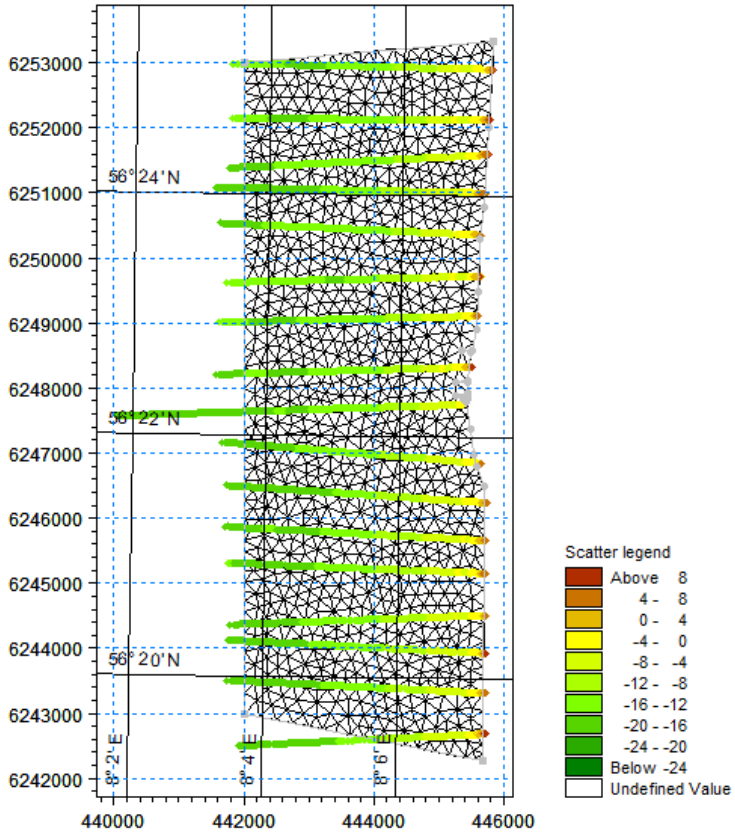


Figure 5.6 Resulting mesh with scatter data from regional lines (2026 elements)

After selecting the scatter data, interpolate the bathymetry using the default settings.

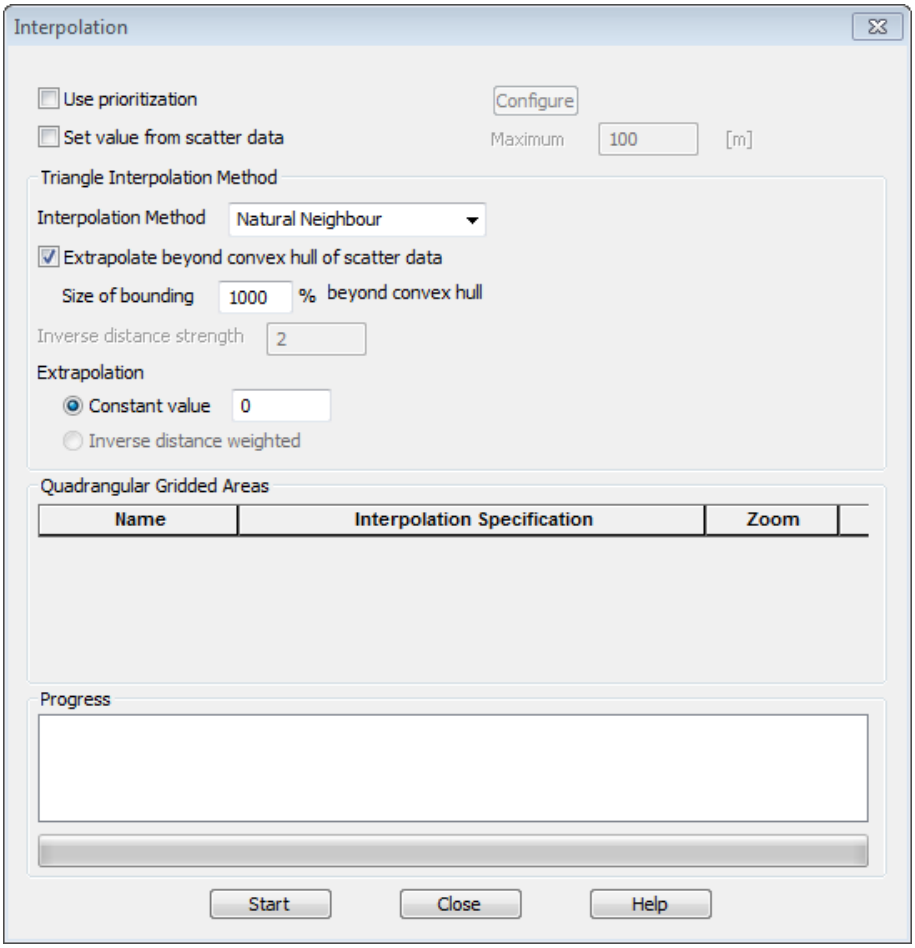


Figure 5.7 Default settings in interpolation

The resulting bathymetry contains values from -20 to +10 m. The analysis show that the maximum time step based on MWL=0m is around 3.7 s and that the limiting elements are near the outer boundary where the water depth is the largest.

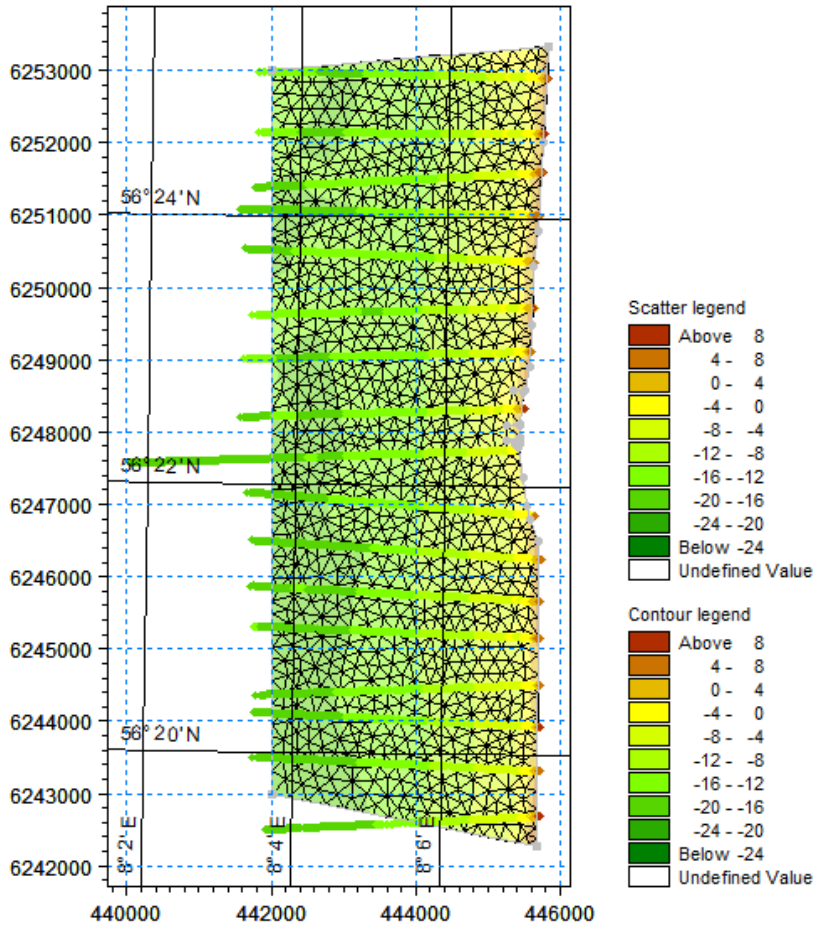


Figure 5.8 Resulting mesh bathymetry including applied scatter data

5.4 Refine Mesh in Shallow Areas

As the waves approach the shoreline the decreasing depth will cause the waves to break. It is possible to refine the mesh, so the mesh elements in the shallower areas decrease with the water depth. This process will not necessarily increase the maximum time step as the outer elements will remain the same.

To refine by depth/gradient you have to have defined a zone before generating the mesh.

Start over by deleting the generated mesh and insert a polygon marker in the domain. Select the polygon marker and right-click to edit the polygon properties. Per default the polygon will be excluded from the mesh, so change the properties to ‘Apply triangular mesh’ and continue the usual way: generate the mesh, smooth the mesh and interpolate the bathymetry.

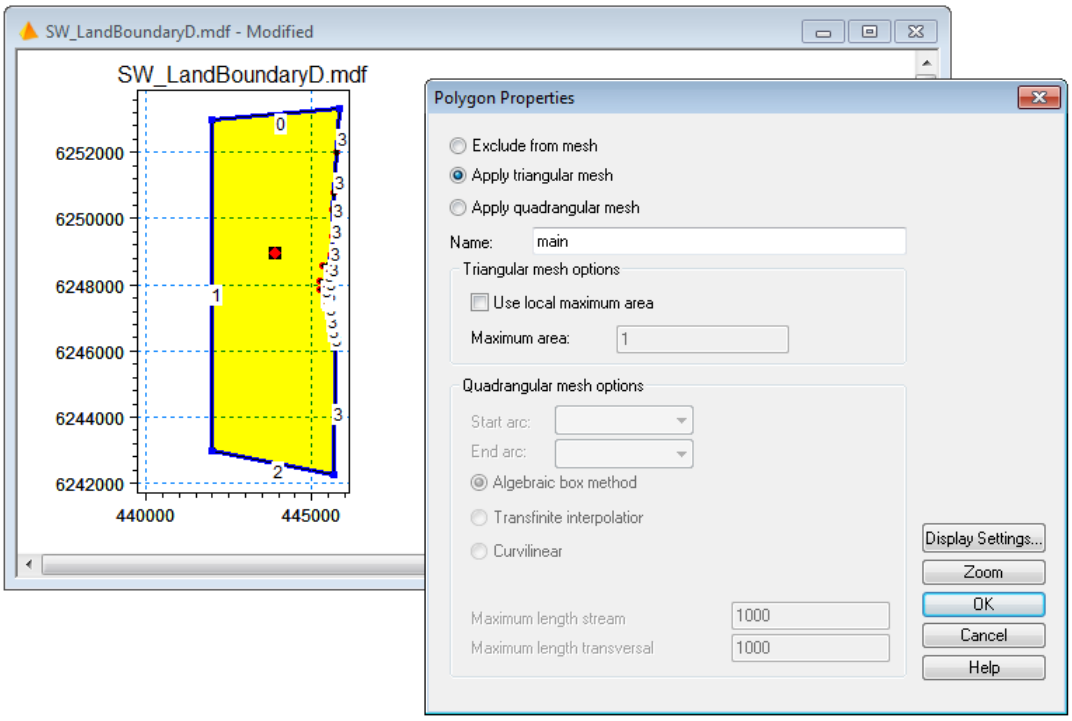


Figure 5.9 Specifying polygon properties for main area

Once the mesh has been generated you can select to refine the mesh by depth or gradient. Per default the relation is linear from the largest value to the smallest, but it is possible to modify the ratio manually.

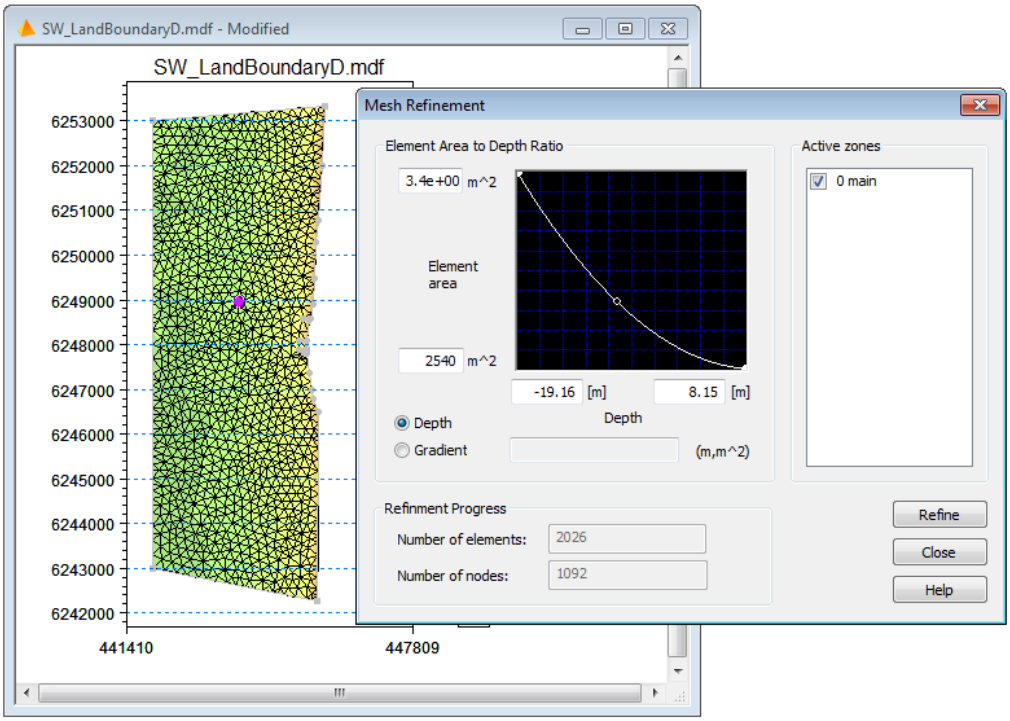


Figure 5.10 Specifying mesh refinement by depth ratio

In the example given the mesh refinement resulted in a mesh with 3699 elements, while maintaining a maximum time step of 3.7 seconds in the Analysis.

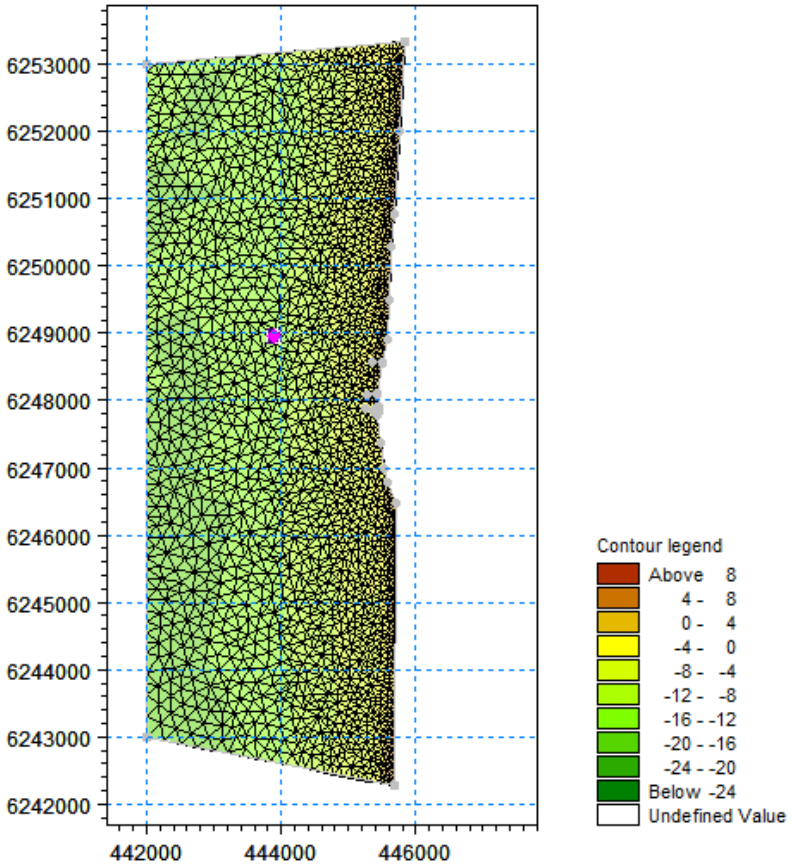


Figure 5.11 Resulting mesh after refinement

5.5 Influence of Mesh Resolution on Simulation Results

To evaluate the effect of the size of the mesh elements on the results the MIKE 21 SW model has been setup for modelling a stationary wave with a height of 3 m and coming from 300 degrees N. Three different meshes were applied in turn: the initial mesh (shown in Figure 4.4), the detail SW mesh (shown in Figure 5.8) and the refined detail SW mesh (shown in Figure 5.11).

The resulting wave fields are shown in Figure 5.12. It can be seen that the overall wave transformation is the same, however the more refined the mesh, the more details can be derived from the results. Thus, it is viable to use a mesh with large mesh elements in areas that are not the subject of close investigation and use finer elements in focus areas.

Note that the simulated wave conditions in the northern part of the area are not similar to the rest, although this should be the case. The difference in the conditions is due to the definition of the northern boundary, which shows the importance of setting up a model where the focus area is well away from the boundary.

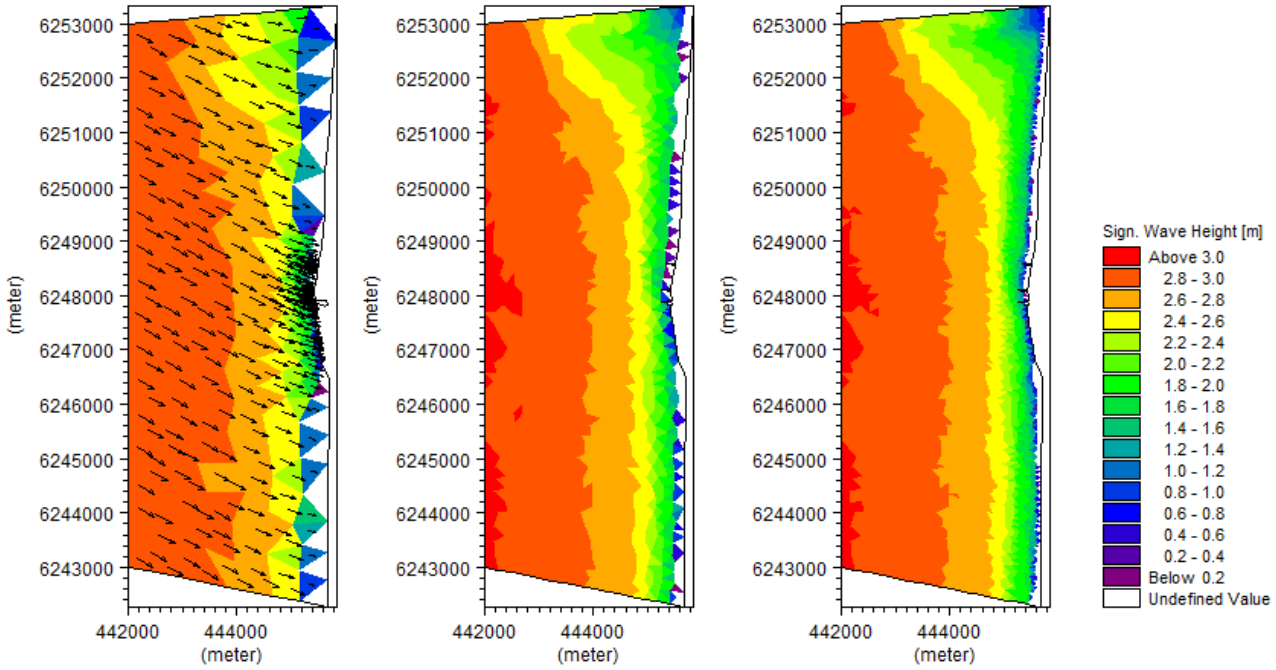


Figure 5.12 Resulting wave field after wave simulation.
Left: initial mesh, Middle: detailed mesh, Right: detailed mesh refined by depth ratio

6 Mesh for Local Wave Simulation

Suppose the aim of the simulation is to investigate the influence of the local bathymetry on the wave conditions close to the harbour entrance.

In this case the details of the land contour and harbour structures is of importance as to the accuracy of the bathymetry in the detail. Also, details in the bathymetry should be resolved in order to evaluate the transformation of the waves in detail. This entails both better scatter data and a more refined mesh in the area.

6.1 Available Data

The land boundary is digitised from map. Several scatter data sets are available, both line surveys and surveys carried out in the harbour entrance.

An overview is given in Figure 6.1.

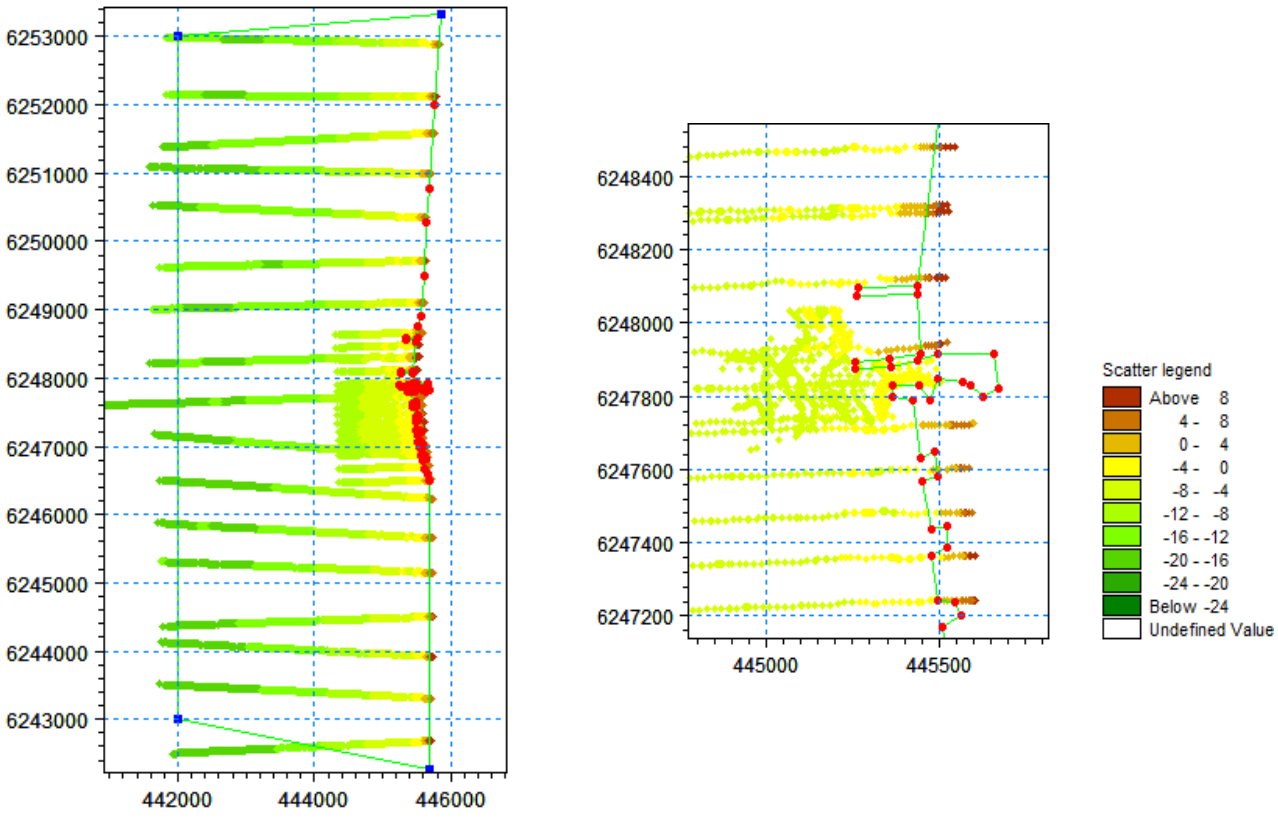


Figure 6.1 Available land and scatter data information.
Conditions around the harbour entrance shown in detail to the right

6.2 Modifying the Land Boundary

The mesh generator setup for the initial resulting mesh is used as a starting point for optimising the mesh.

The land contours away from the harbour entrance can be simplified as this will not affect the local wave conditions near the entrance. However, the land contours representing the harbour from the first digitisation were quite coarse.

One of the scatter data sets were derived by digitising a survey map, created by the local harbour authorities in 1996 (see Figure 6.2). This map contains details of the harbour layout that can be used to improve the definition of the land contour in the local area.

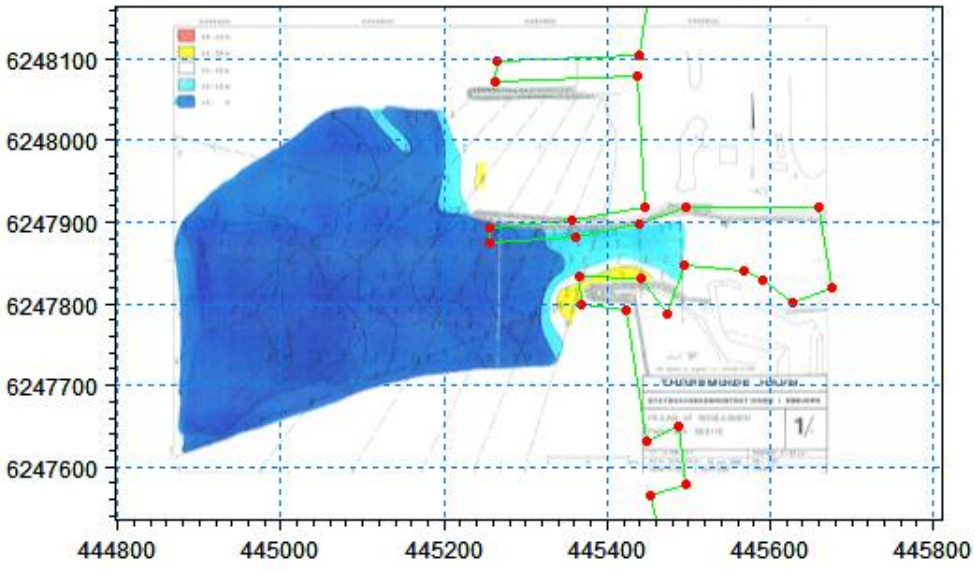


Figure 6.2 Bathymetric survey shown as background map to digitised land boundary

By moving and adding vertices to the land boundary it is possible to obtain a better resolution of the harbour entrance as shown in Figure 6.3. When you are satisfied with your modifications it may be a good idea to select the land boundary line and export the arc to a xyz file for possible use later on.

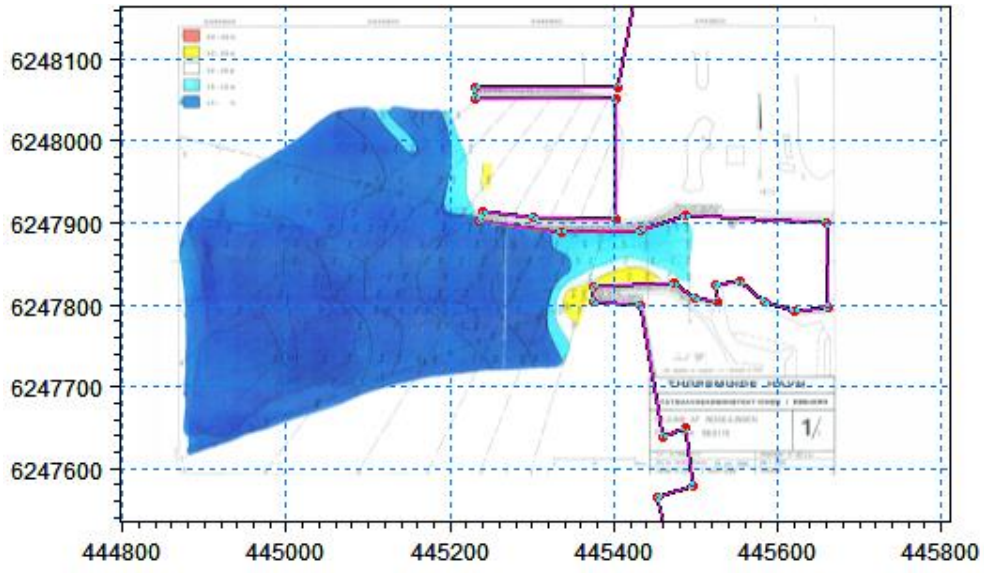


Figure 6.3 Bathymetric survey shown as background map to edited land boundary

Note that the land boundary is defined with as few vertices as possible in order to avoid small elements.

6.3 Using Polygons in Mesh Generation

In the area near the harbour the mesh resolution should be detailed, whereas the deeper areas will do fine with a coarser mesh resolution. Apart from the general refinement tools applied in the previous chapter, it is possible to control the mesh resolution in certain areas by defining polygons where the local maximum element area can be defined.

A polygon is an enclosed area surrounded by arcs. By adding a polygon marker inside the polygon you may define the mesh resolution for the individual polygon. The node point density within a polygon may be defined by selecting the polygon marker; right click the indicated area and select Properties in the dropdown menu (cf. Figure 6.4). This will open a dialogue where you can define the type of the mesh element (triangular/quadrangular) and the maximum element area within the polygon (cf. Figure 6.5). Note that if you do not define a maximum element area in this dialogue, the maximum element area will be defined by the triangular mesh element options in the Mesh Generation dialogue.

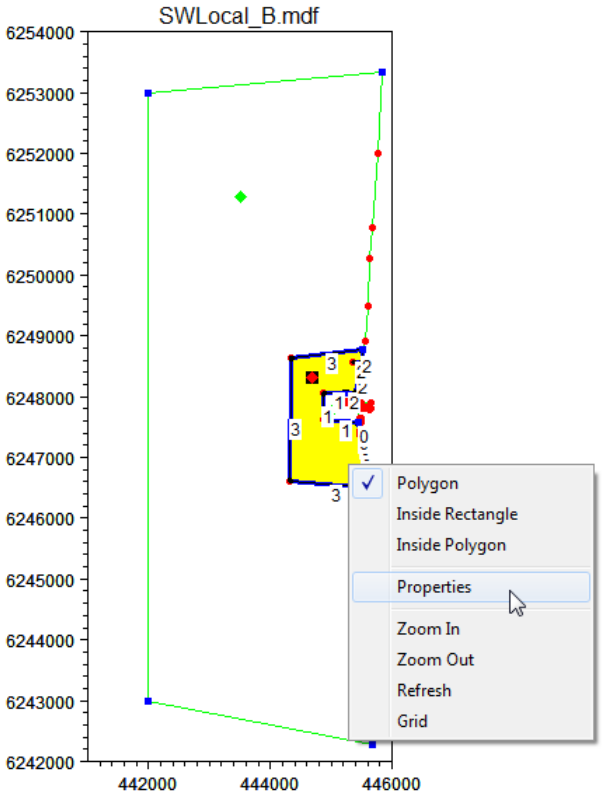


Figure 6.4 Selection of polygon area to define properties

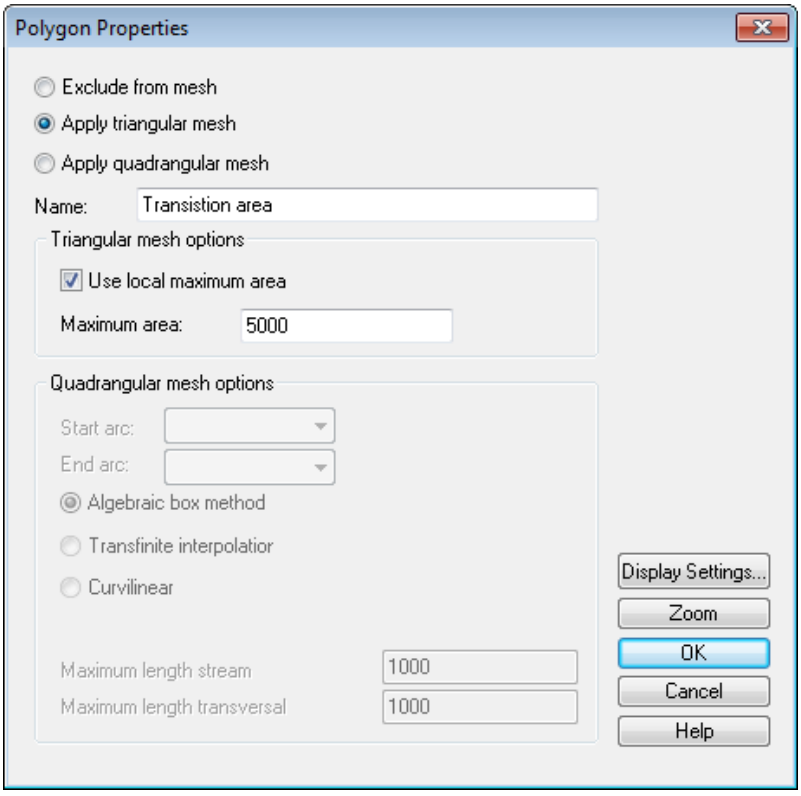


Figure 6.5 Definition of the properties of a defined polygon

The focus of the present simulation is on the area just outside the harbour, why the mesh elements should be the smallest in this area. To ensure a smooth transition between large elements in the deeper waters and small elements near the harbour, a third polygon is defined as a transition area. The polygons can be defined to follow the outline of the different scatter data sets as shown in Figure 6.6. However, this is not necessary as the generation of the mesh and the interpolation of the bathymetry are two separate processes.

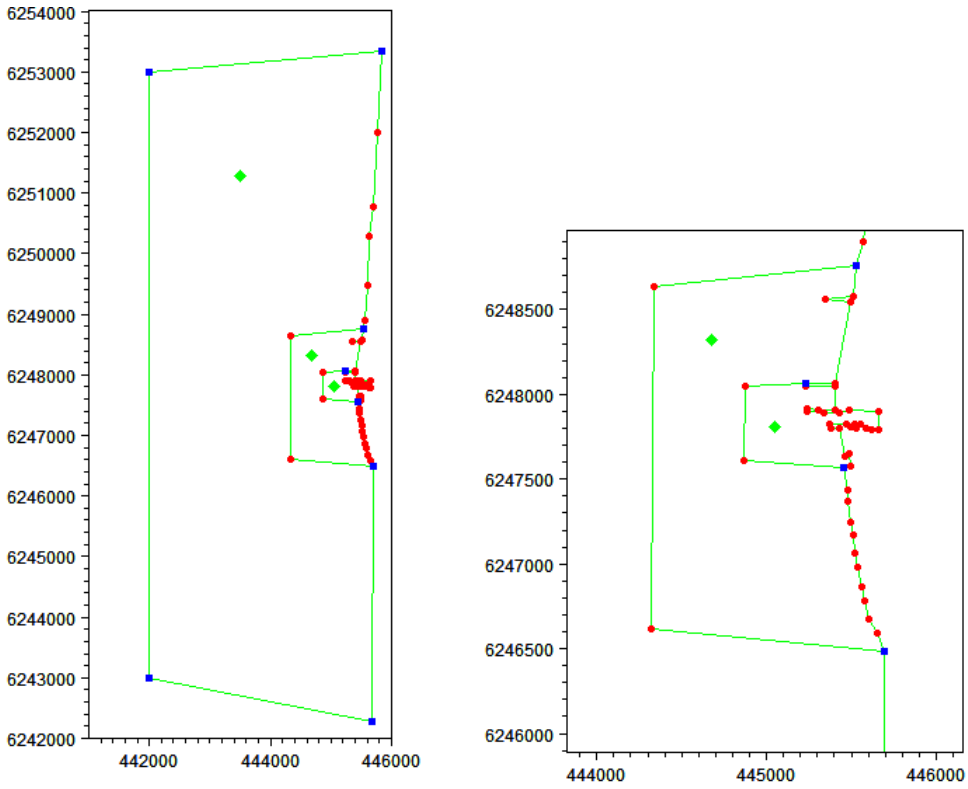


Figure 6.6 Definition of polygons around harbour entrance

In this case the outer area is defined by overall settings with the maximum area set to 30000 m^2, the transition area is defined by a maximum element area of 5000 m^2 and the inner area is defined by a maximum element area of 1000 m^2. Area wise the scaling between different transitions areas should be in the range 4 to10 giving a length scaling approximately between 2 and 3.

This gives a resulting mesh where the three polygon areas have distinctly different mesh resolutions as shown to the left in Figure 6.7.

Please note that during the process of smoothing, the mesh elements there is an option to leave mesh nodes at arcs untouched. Selecting this will mean that the internal arc defining the polygon will be clearly seen in the resulting mesh. Thus, in this case the option should be unchecked in order to get a smooth transition between the areas with different resolution.

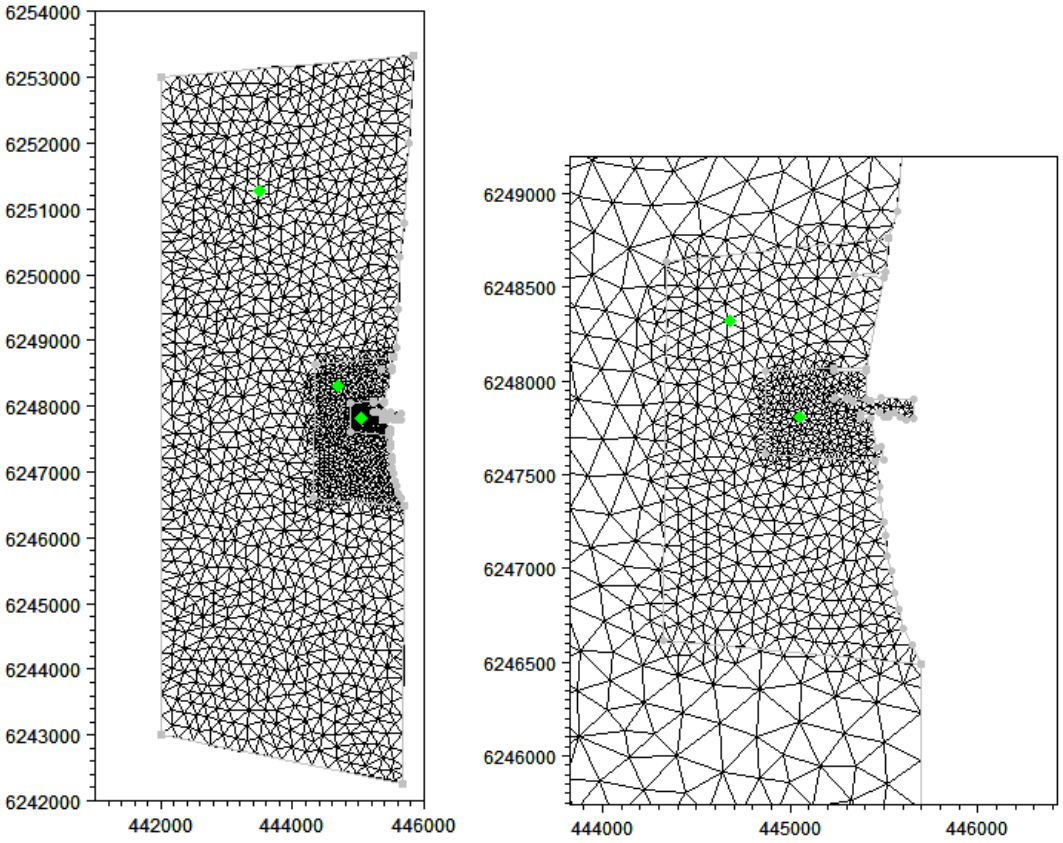


Figure 6.7 Generated mesh with using polygons (3219 elements).
Left: unsmoothed, Right: detail after smoothing

6.4 Prioritise Scatter Data

In the area near the harbour a number of different scatter data sets are available. It is possible to add several data sets to the setup in mesh generator and then choose to view and/or use only some of them in the mesh interpolation.

When adding several data sets to a setup some of the data may be obtained in the same area. By just adding all data to the interpolation the data points are considered equally valid and the resulting bathymetry will be derived as a mean of the available data. However, often it would be preferred that some areas are mostly affected by one particular data set. To enforce this prioritisation, prioritisation areas can be included in the mesh generation, such that the weight of each data set can be explicitly defined.

In the present example prioritisation areas are defined to cover the areas with overlaying data as shown in Figure 6.8. Note that the prioritisation areas do not have to correspond to the polygons used for the mesh generation.

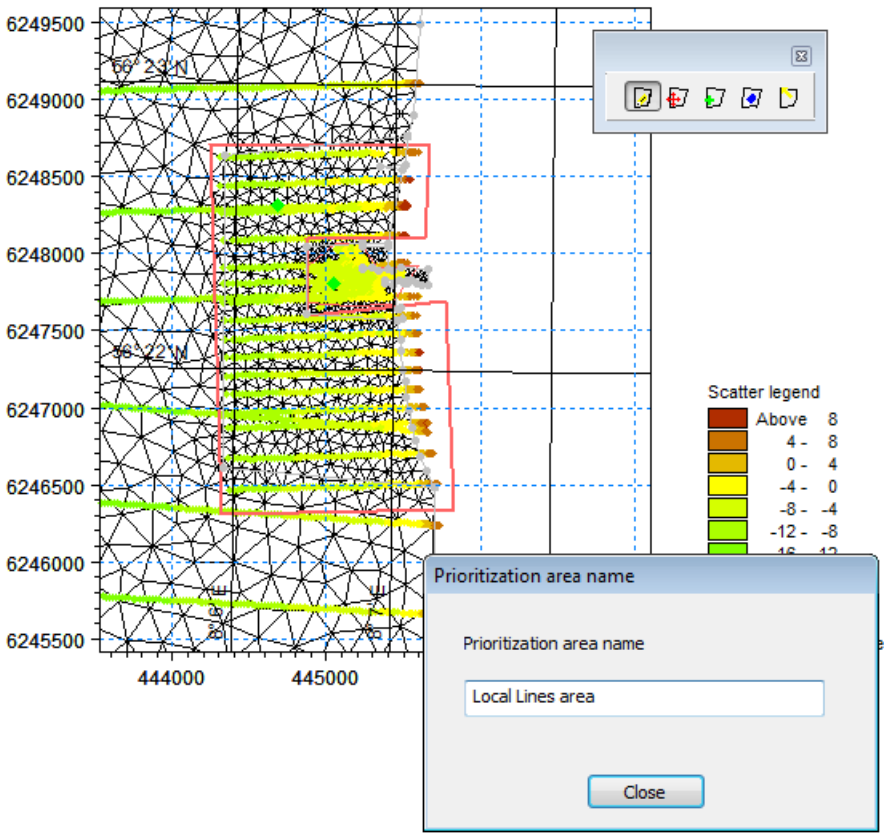


Figure 6.8 Specification of prioritisation area

Once the prioritisation areas have been defined it is possible to prioritise the scatter data in the different prioritisation areas. This is done by defining different weight combinations for the available scatter data sets, and then applying one of the defined weight combinations to the individual prioritization area as shown in Figure 6.9.

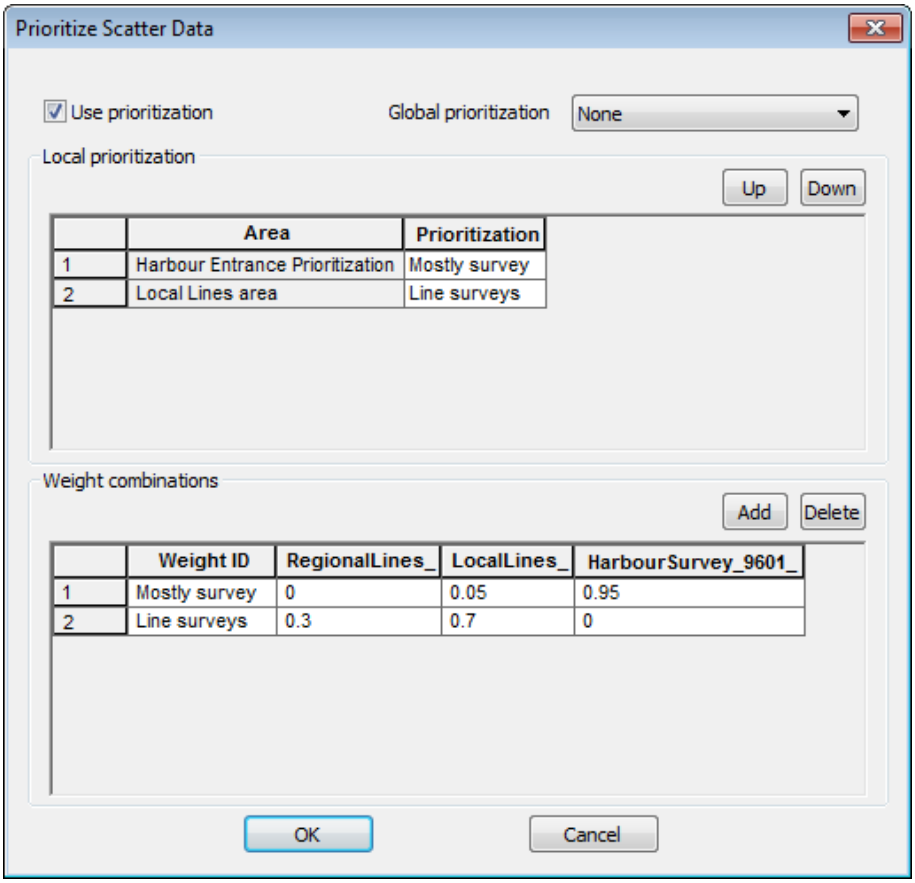


Figure 6.9 Specification of prioritised scatter data in different areas

In this case, for example, the bathymetry in the harbour entrance is found by 95% weight on the harbour survey and 5% weight on the local line survey. The regional line survey data is not considered in this area.

Using this in the interpolation of the bathymetry (remember to enable ‘Use prioritization’ in the dialogue), it can be seen that the consistency of the depth contours near the harbour entrance is better when the harbour survey is dominant in the area than if all data sets has the same weight in the interpolation.

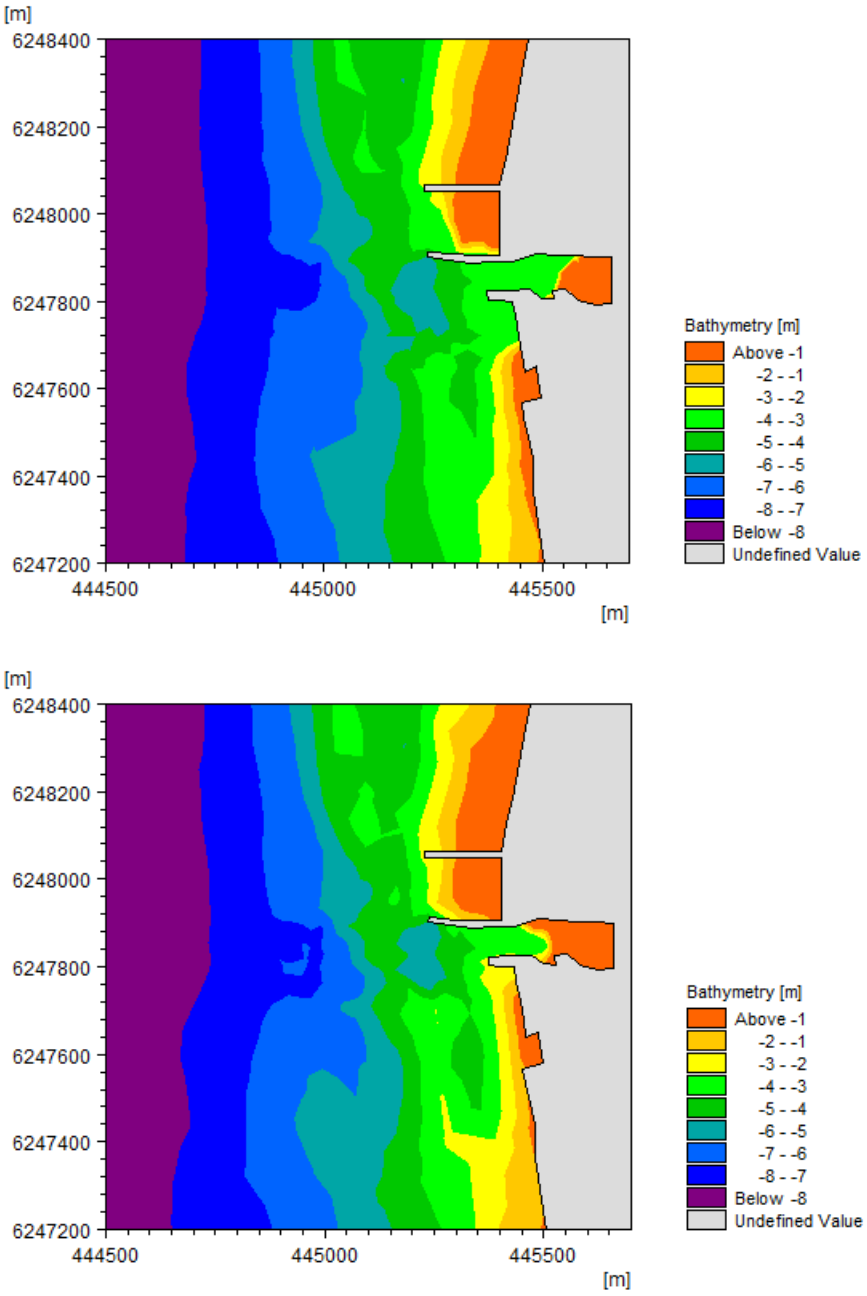


Figure 6.10 Resulting mesh bathymetry
Upper: including prioritisation, Lower: excluding prioritisation

6.5 Updating Mesh Bathymetry Using New Scatter Data

In case new scatter data must be considered it is not necessary to eliminate the existing scatter data from the model setup. The change may be effectuated by modifying the weight on the different scatter data sets, see Figure 6.11.

In this case a new harbour survey was conducted after a storm period, which caused a bar to develop in front of the harbour entrance, see Figure 6.12.

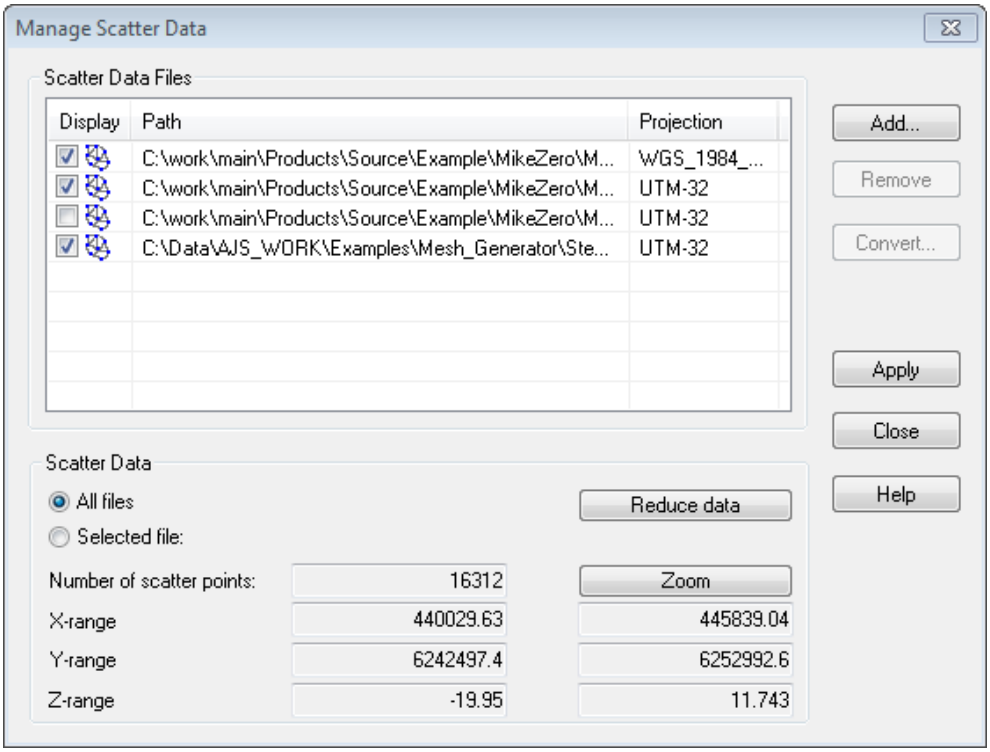


Figure 6.11 Adding a new data set of scatter data. Display of old scatter data set is disabled

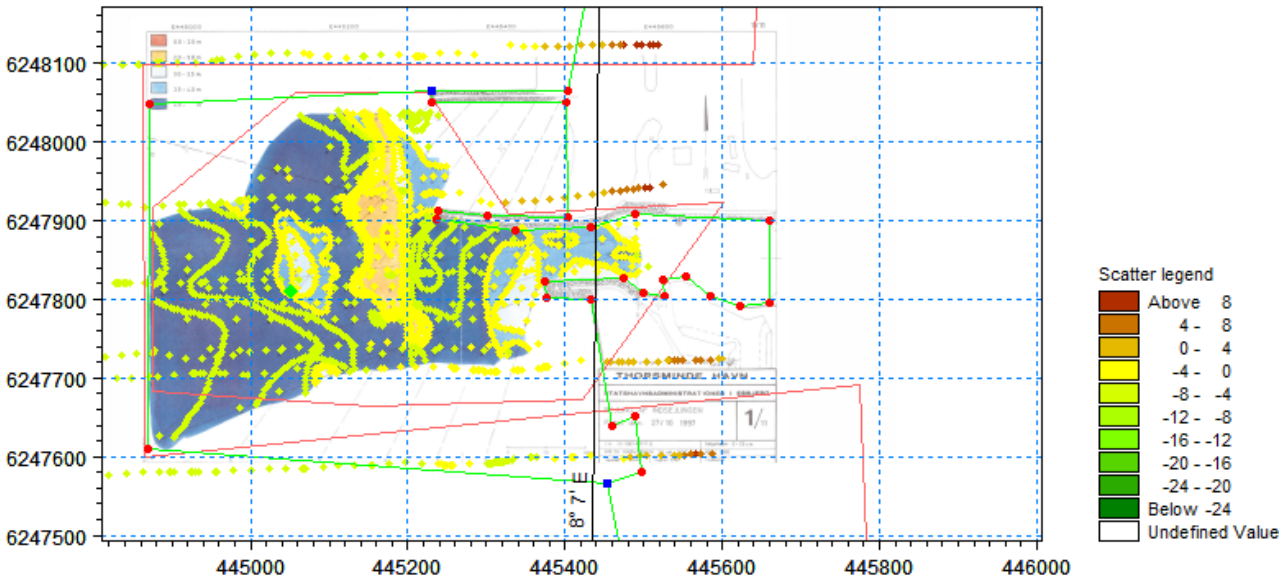


Figure 6.12 New harbour survey scatter data including survey image as background layer. Red lines indicate existing prioritisation areas

A new mesh bathymetry is now to be generated based on this new harbour survey, but keeping the existing line surveys. By modifying the weight combinations in the existing prioritisation areas or creating a new weight combination to apply in the prioritisation a new mesh bathymetry can be generated. Note that even if a scatter data set is not displayed, it may still affect the interpolation.

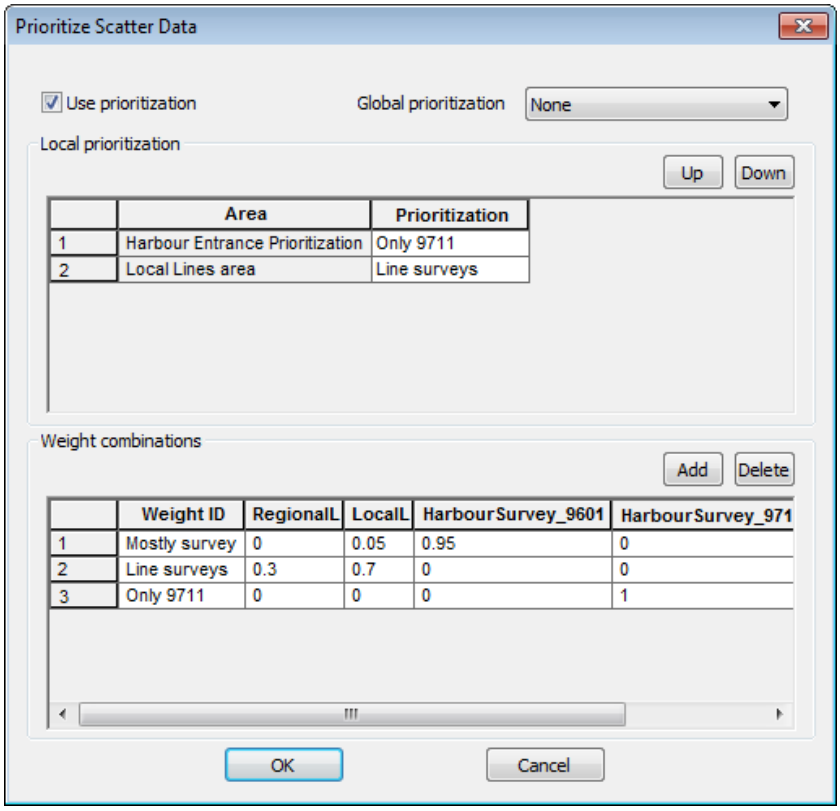


Figure 6.13 Prioritisation settings using new prioritisation for harbour entrance

In the present setup the new scatter data set is the only one used in the interpolation of the bathymetry within the harbour entrance prioritisation area. The resulting mesh bathymetry is shown in Figure 6.14.

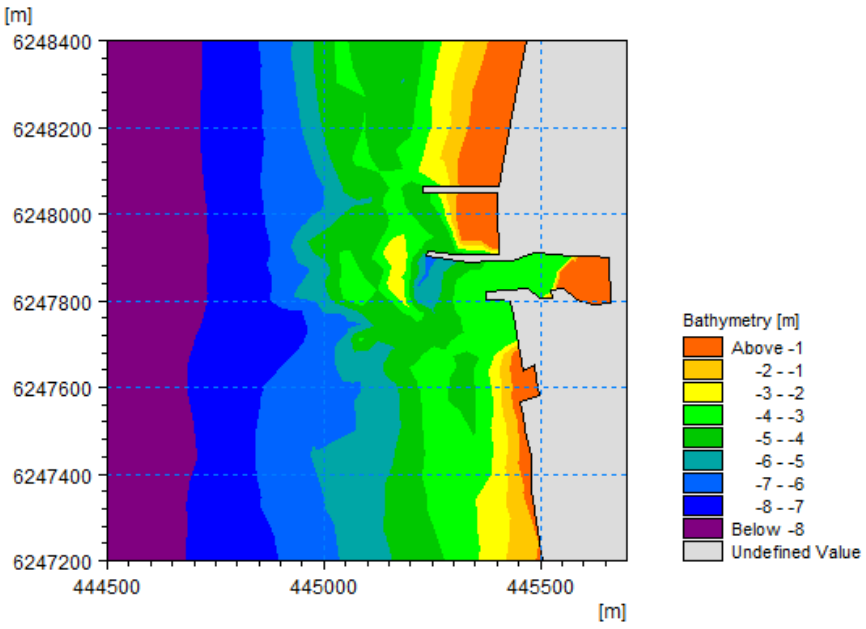


Figure 6.14 Resulting mesh bathymetry using harbour survey from 1997 after a storm

6.6 Influence of Bathymetry on Simulation Results

To evaluate the influence of the different bathymetries the Spectral Wave Model is rerun with the newly generated bathymetries. The result originating from the 1996 bathymetry is shown in Figure 6.15 and the result from the 1997 bathymetry is shown in Figure 6.16.

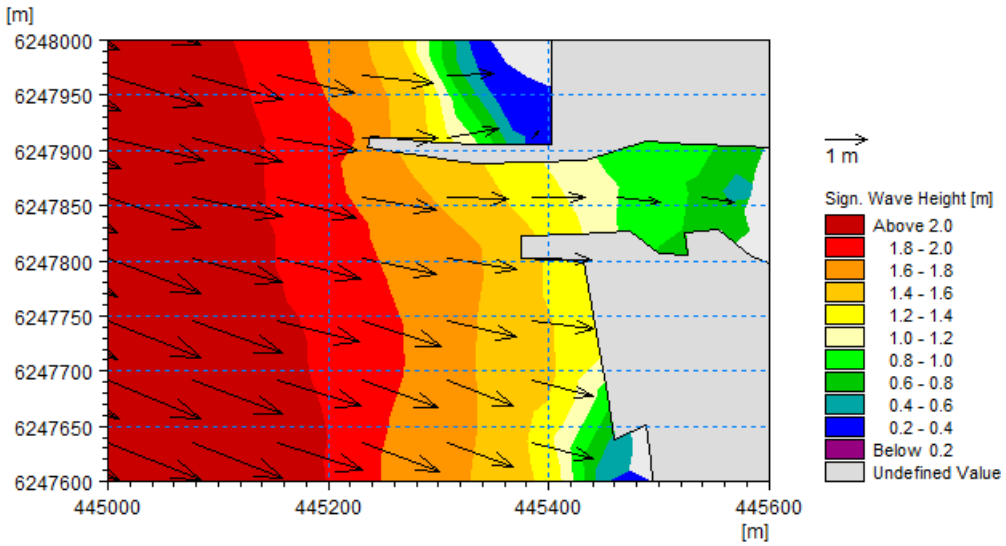


Figure 6.15 Resulting wave field for 1996 survey

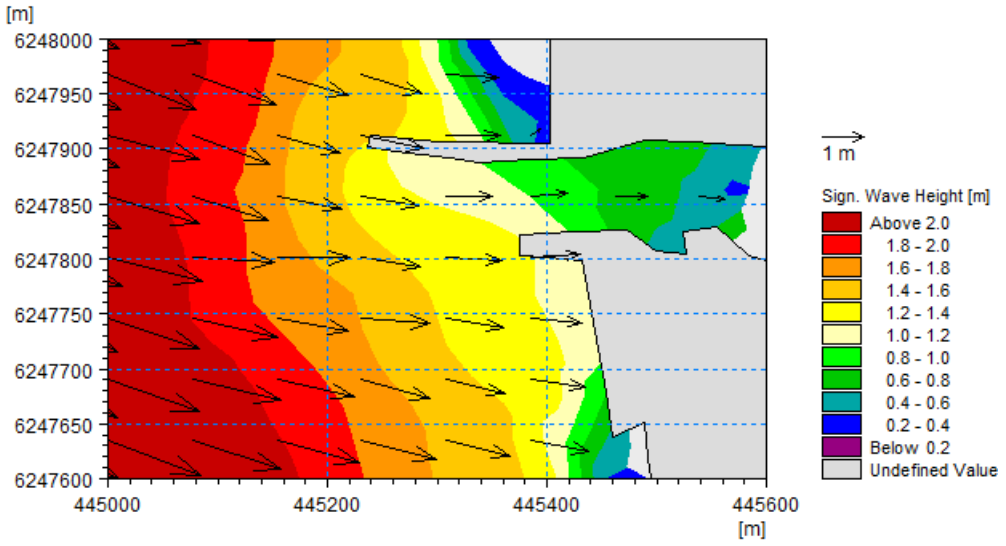


Figure 6.16 Resulting wave field for 1997 survey

From both surveys it can be seen that the presence of the bar in front of the harbour has an influence on the wave field.

Likewise a MIKE 21 HD FM model is setup to simulate a current field created by a constant water level gradient of 0.1 in the north-south direction. The results from using the two different bathymetries are shown in Figure 6.17 and Figure 6.18.

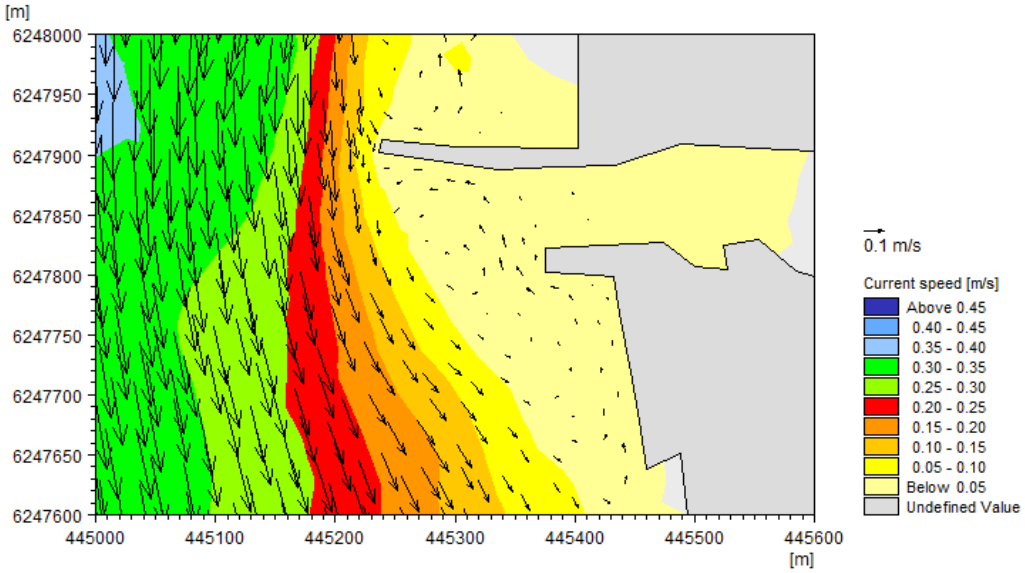


Figure 6.17 Resulting flow field using the 1996 survey

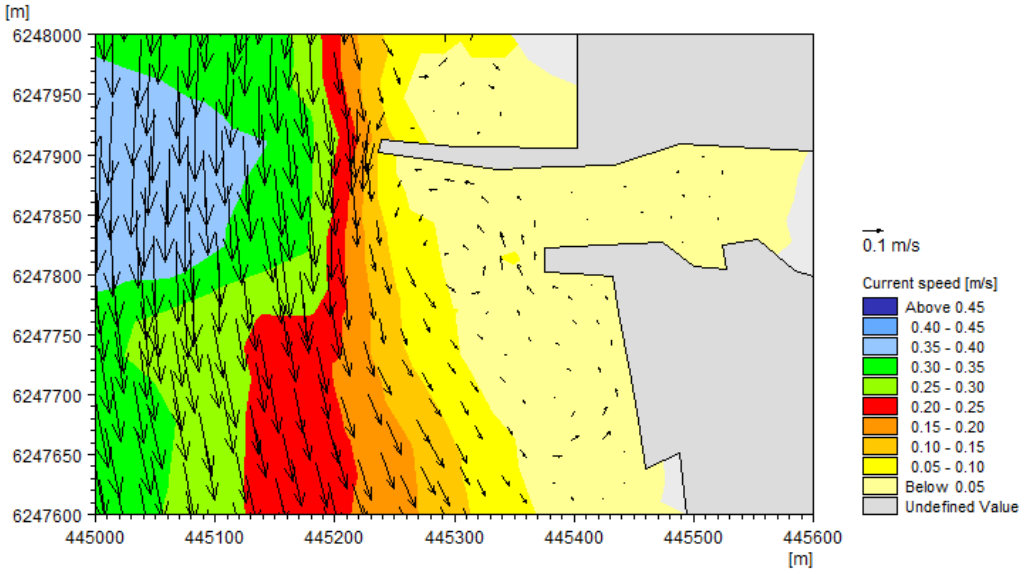


Figure 6.18 Resulting flow field using the 1997 survey

It can be seen that the presence of the bar will cause the flow field downstream the harbour entrance to diverge once the bar ends and the water depth increases.

7 Mesh for Longshore Current Simulation

Suppose the aim of the simulation is to investigate the cross-shore distribution of the wave properties and the longshore current profile, due to an incoming wave on a nearly uniform coastline. In this case the wave and the current conditions would be nearly constant in the longshore direction, but highly varying across the shore.

The details across the surf zone are very important and as such the mesh elements must be sufficiently small to resolve the processes of the wave transformation. It is, however, not necessary to have a fine resolution in the longshore direction as the conditions will be nearly constant from one cross-section to the next.

Note that this type of condition is often similar to the requirements for modelling river flow.

7.1 Defining Polygons for Customisation

As the cross-shore section must be resolved with a high resolution in order to describe the wave generated long shore current, the cross shore resolution should normally cover the breaking zone with at least 5 to 10 elements cross shore. The resulting mesh may end up containing very many elements by using the default mesh generation with triangular elements as there is a limit on the minimum angle in a triangular element. It is, however, possible instead to define the mesh by quadrangular elements (or a combination of the two types). Using quadrangular elements entails the possibility of having a fine resolution across the profile and a coarser resolution along the coastline, thus reducing the total number of elements.

Please note that the elongated mesh elements will favour the flow direction along the element. Hence, in case of divergence in the flow triangular elements will be a better choice.

The overall domain area used in the initial mesh generator setup can be used as a template, and the bathymetry values are derived from the regional lines.

The resulting mesh will be used to model south-westerly waves with focus on the coastline north of the harbour entrance. Therefore, in order to define a finer mesh resolution near the focus area, a number of polygons should be defined in order to optimise the solution as shown in Figure 7.1.

As the details along the shoreline south of the harbour will not influence the wave transformation and the longshore currents north of the harbour, the land boundary is simplified in order to avoid unnecessary small elements.

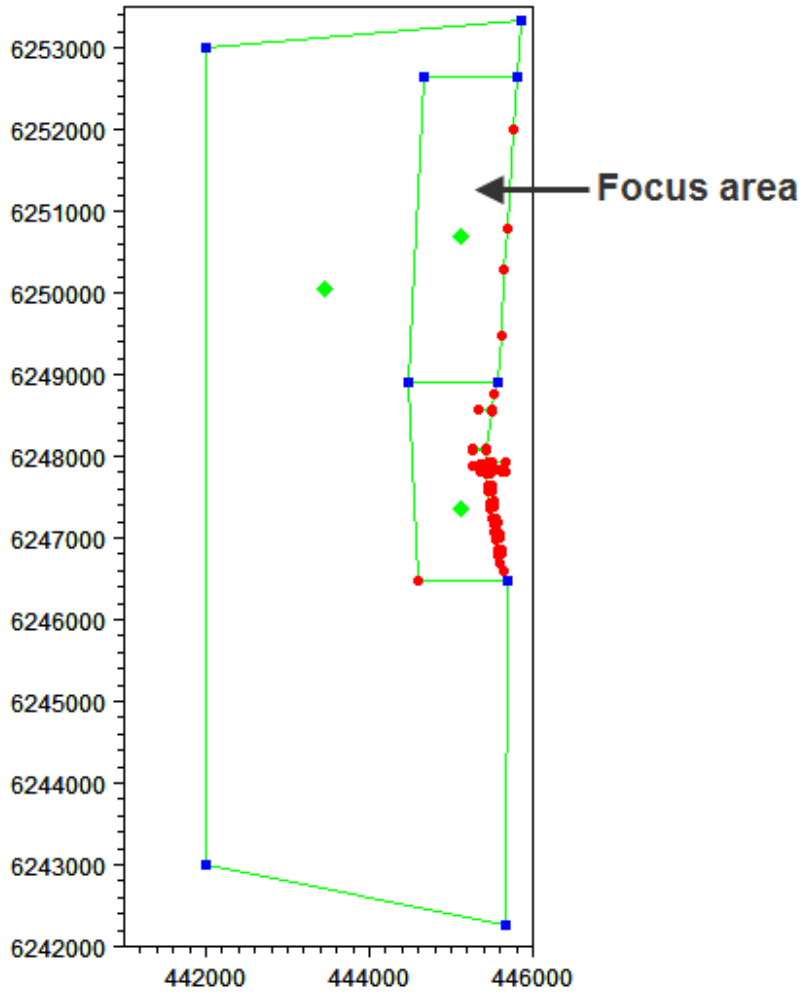


Figure 7.1 Definition of polygon areas

As in the previous cases, the outer area should be to be defined by a triangular mesh with a maximum area of 30000 m^2. To ensure a smoother transition of the longshore currents into the focus area, the southern polygon area will be defined by a triangular mesh using a smaller maximum area, e.g. 5000 m^2. The polygon for the focus area will be defined by quadrangular elements.

7.1.1 Defining quadrangular elements

The necessary information for defining an area with quadrangular elements is different from an area with triangular elements. For quadrangular elements, the related polygon must be defined by at least 4 individual arcs. Two of these arcs are selected to define the start and the end of the rectangular section, and these arcs must not contain any vertices.

To define an area with quadrangular elements, right-click the polygon marker in the display and press Properties (cf. Figure 7.2).

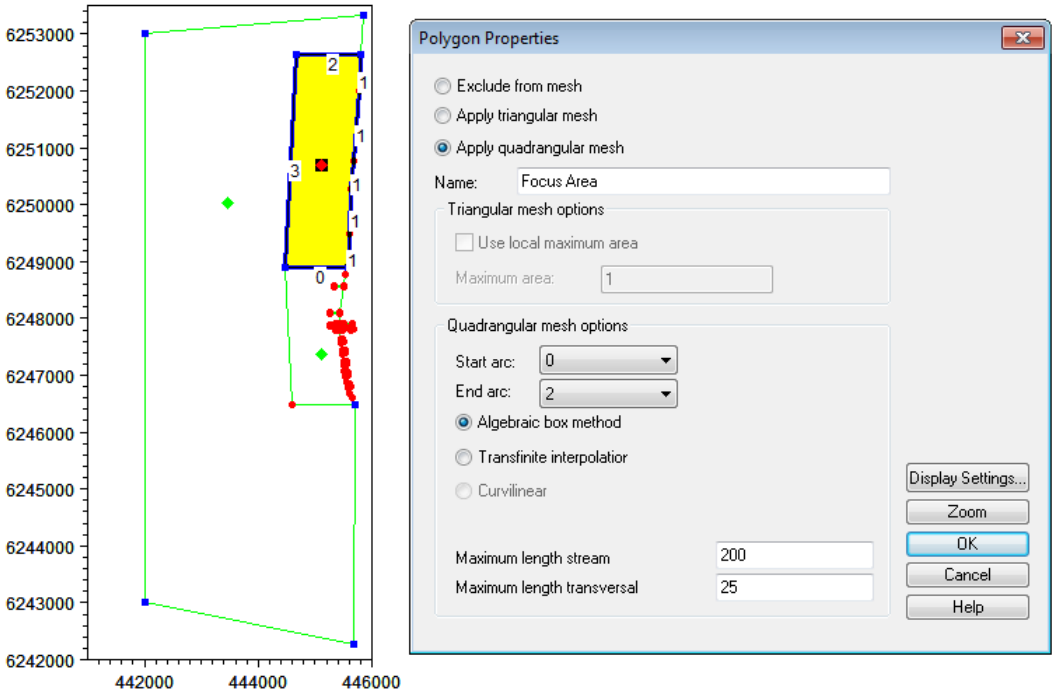


Figure 7.2 Definition of quadrangular elements

First, select ‘Apply a quadrangular mesh’. This will enable the quadrangular mesh options where you select the start arc and the end arc number. Note that these numbers are found internally by the programme and do not relate to the code value of the arc.

Secondly, define the maximum length along the stream and transverse the stream to e.g. 200 m and 25 m, respectively. This is a limit of the mesh element area, similar to the value given for the maximum triangular area. Usually, the length along the stream is larger than across. However there may be cases where it is more feasible to define a stream in the ‘short’ direction.

Finally, press OK to save the definition of the quadrangular polygon area and generate the mesh. During the generation of the mesh the quadrangular mesh elements will be distributed uniformly within the associated polygon.

The resulting mesh could look as shown in Figure 7.3.

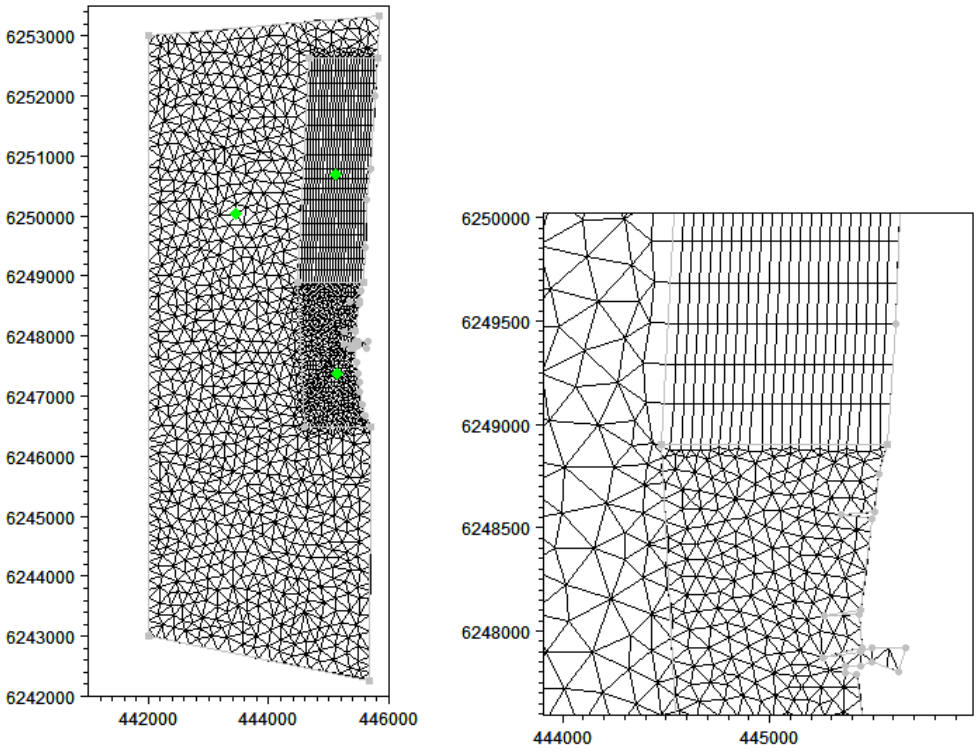


Figure 7.3 Resulting mesh (2943 elements). The figure to the right shows detail after smoothing

7.1.2 Interpolating bathymetry

The bathymetry values are to be based on the measured regional lines. These lines are densely populated across the beach but carried out with some distance along the beach, cf. Figure 7.4.

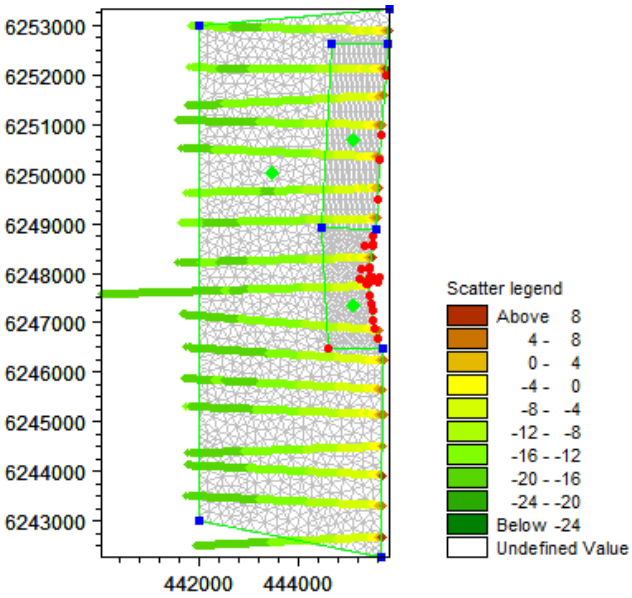


Figure 7.4 Scatter data from regional measuring lines

This kind of scatter data requires careful consideration with respect to the type of interpolation. When interpolating scatter data in quadrangular element areas, there are three types of interpolation methods as shown in Figure 7.5.

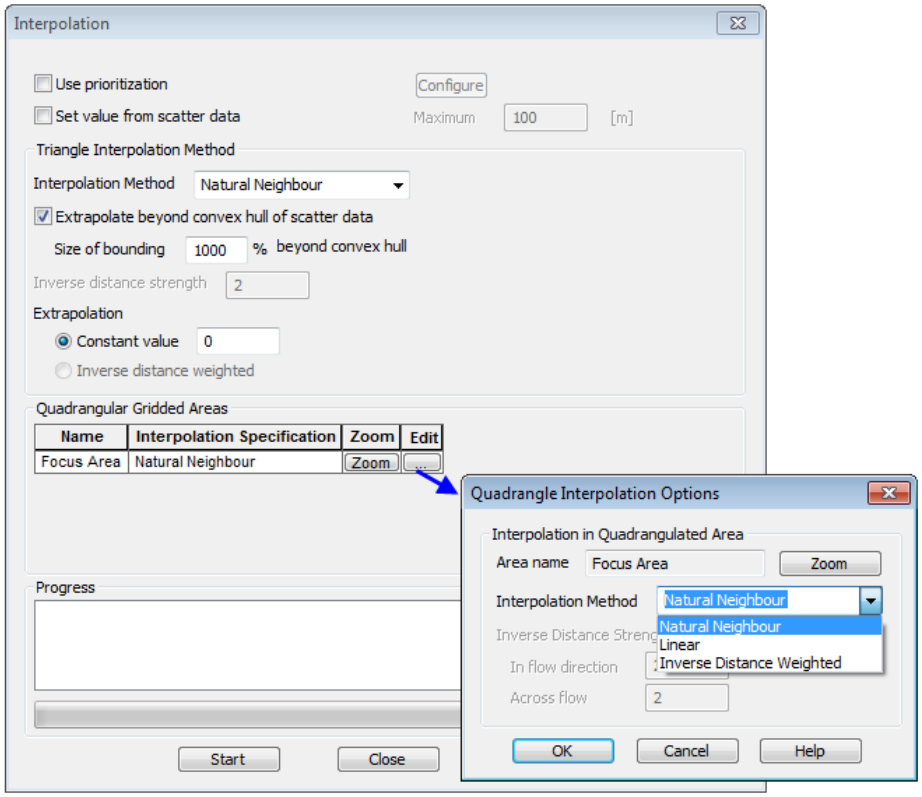


Figure 7.5 Definition of quadrangular interpolation options

The Natural neighbour and Linear options are similar to the options for the triangular elements. The Inverse Distance Weighted option enables the optimal use of scatter data where the points are not distributed uniformly in the area: by setting a ‘strength’ value for the scatter data points in the longshore and cross-shore direction respectively, it is possible to obtain a kind of prioritisation of the data in the interpolation process.

In the mesh shown in Figure 7.4, the resolution of the quadrangular grid is sufficiently coarse to limit the influence of the choice of interpolation. However, in case the resolution is fine and there are several mesh elements between the measured lines, the choice of interpolation becomes important. This is the case if the quadrangular element area is as detailed as shown in Figure 7.6.

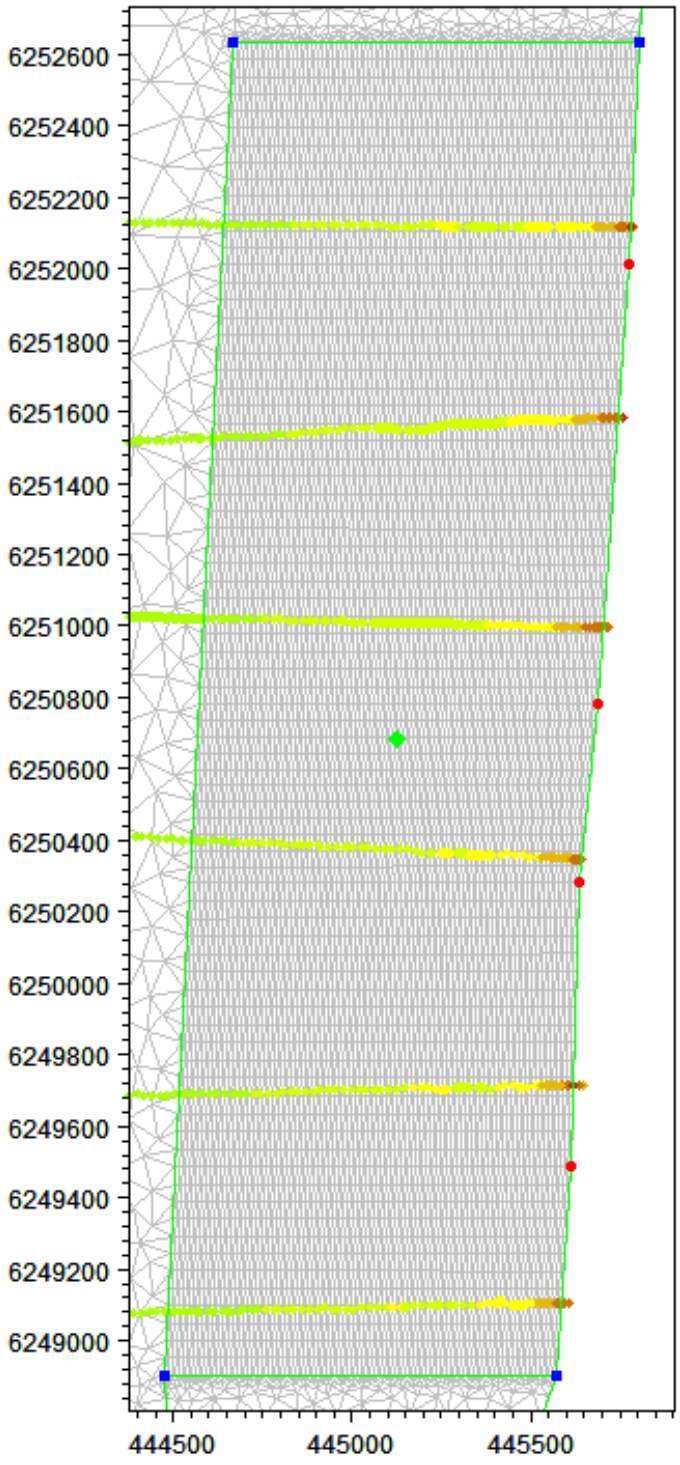


Figure 7.6 Detailed quadrangular mesh area with measured regional scatter data lines

Figure 7.7 - Figure 7.10 shows the results of the different interpolation methods for a part of the focus area when using the detailed mesh shown in Figure 7.6.

Please note that the depth contours in between the measured lines show irregularities for linear and natural neighbour interpolation. By applying the inverse distance method the default weight values gives unrealistic results, but by applying a different weight factor for

the x and y direction, respectively, the expected symmetry of the natural bathymetry along the beach can be reproduced.

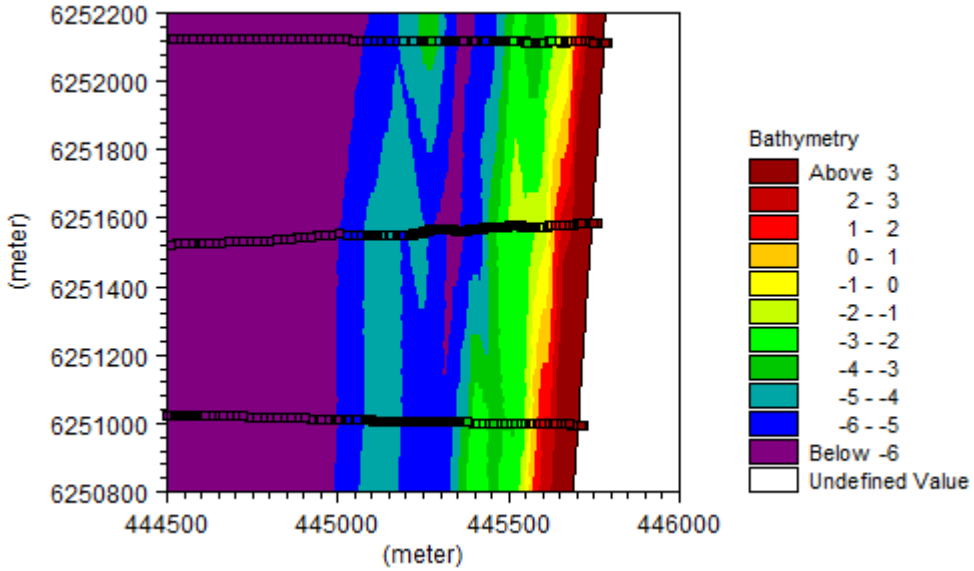


Figure 7.7 Interpolated bathymetry using linear interpolation

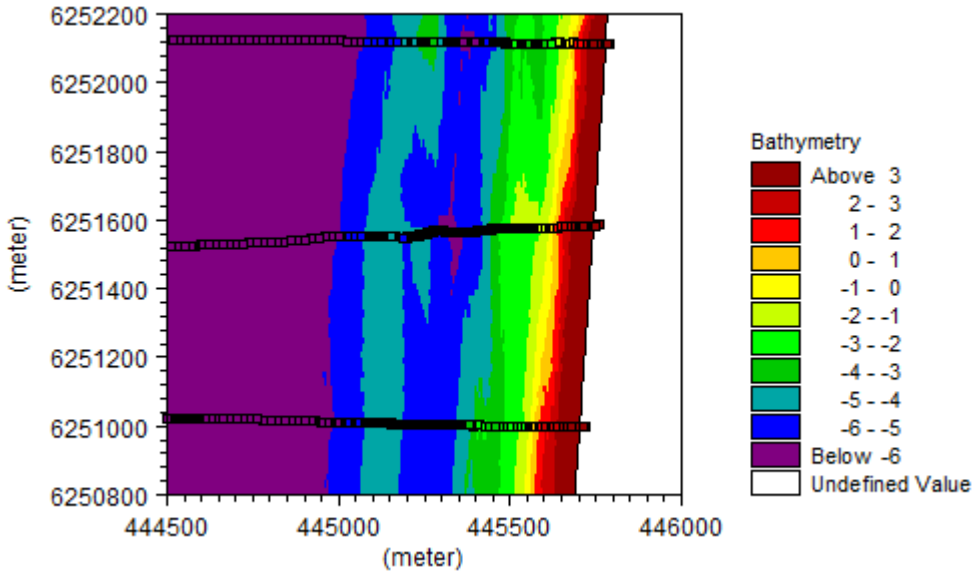


Figure 7.8 Interpolated bathymetry using natural neighbour interpolation

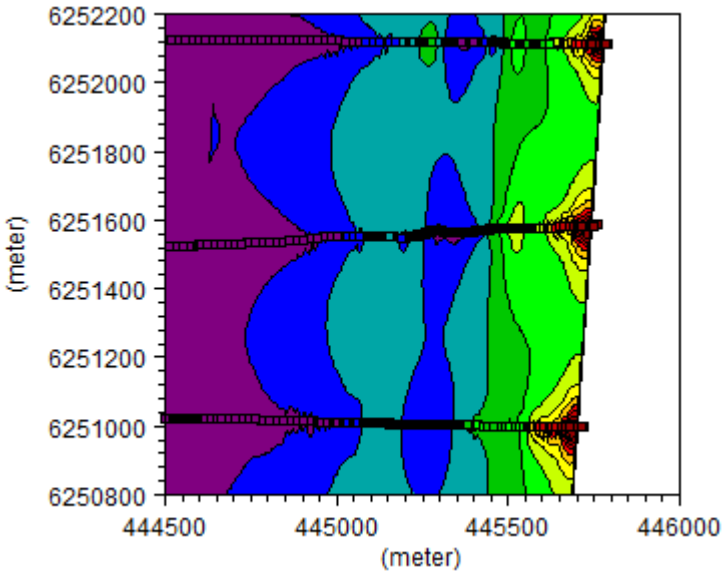


Figure 7.9 Interpolated bathymetry using Inverse distance weighted interpolation. Strength value in flow direction: 2, and strength value transverse: 2

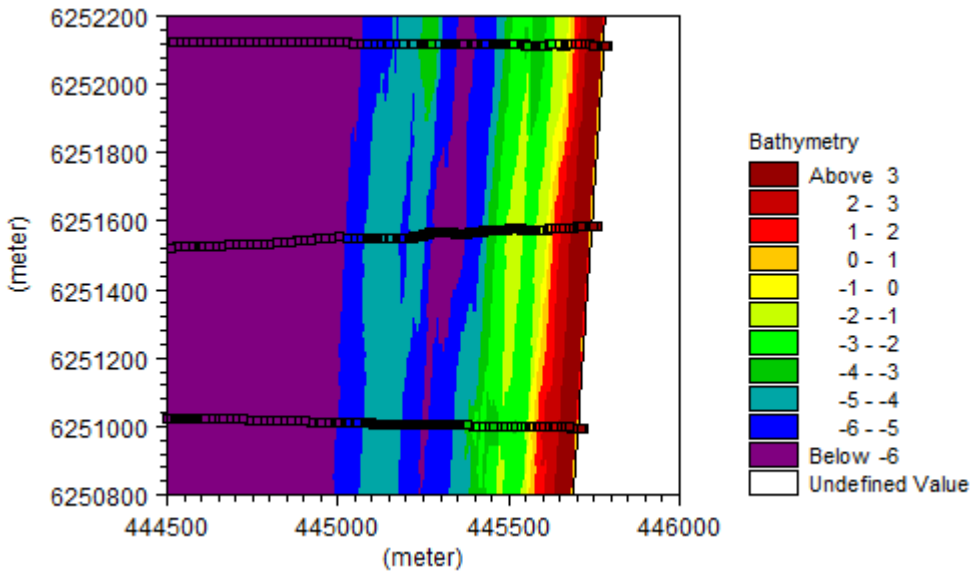


Figure 7.10 Interpolated bathymetry using Inverse distance weighted interpolation. Strength value in flow direction: 2, and strength value transverse: 20

7.2 Neighbouring Quadrangular Element Areas

It is possible to generate a mesh with adjacent different quadrangular mesh element areas when they share one of the definition arcs, i.e. are connected in the longitudinal direction. In case the quadrangular elements areas cannot be defined like this, it is necessary to generate a temporary mesh with a small band of triangular elements to divide the two areas and then modify the mesh elements manually to connect the quadrangular areas.

The example in Figure 7.11 shows how to create complex quadrangular mesh areas: Assume that you south of the harbour entrance require 3 areas with quadrangular elements to be located adjacent to each other. The mesh elements within Area A must have a maximum length of 200 m and a width of 25 m (across the beach). The mesh elements in Area B must have a maximum length of 200 m and a maximum width of 100m. The mesh elements in Area C must have a maximum length of 100 m.

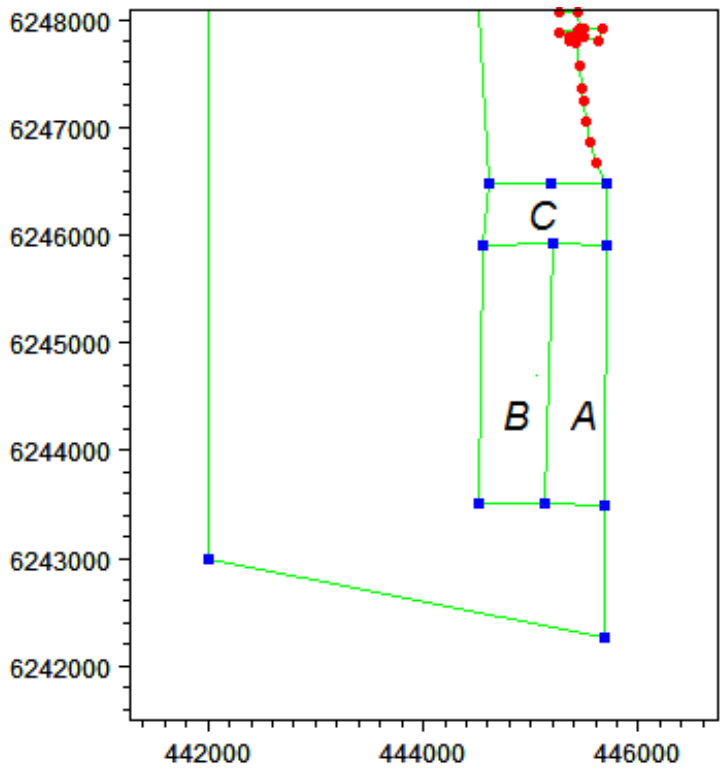


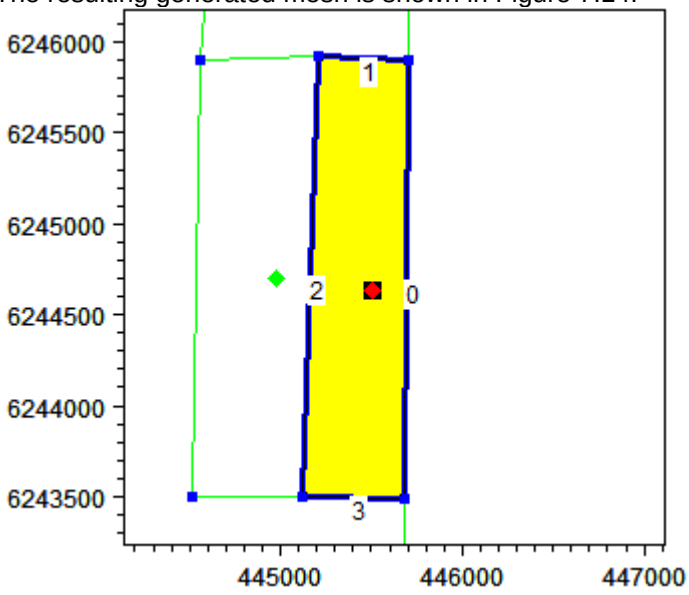
Figure 7.11 Mesh Generator setup with 3 areas indicated for rectangular elements

7.2.1 Two adjacent areas

First define the polygons for Area A and Area B as shown in Figure 7.12 and Figure 7.13.

Please note: It is only possible to connect two quadrangular polygon areas at the start or end arc. Thus the maximum length transversal must be the same for the two areas.

The resulting generated mesh is shown in Figure 7.14.



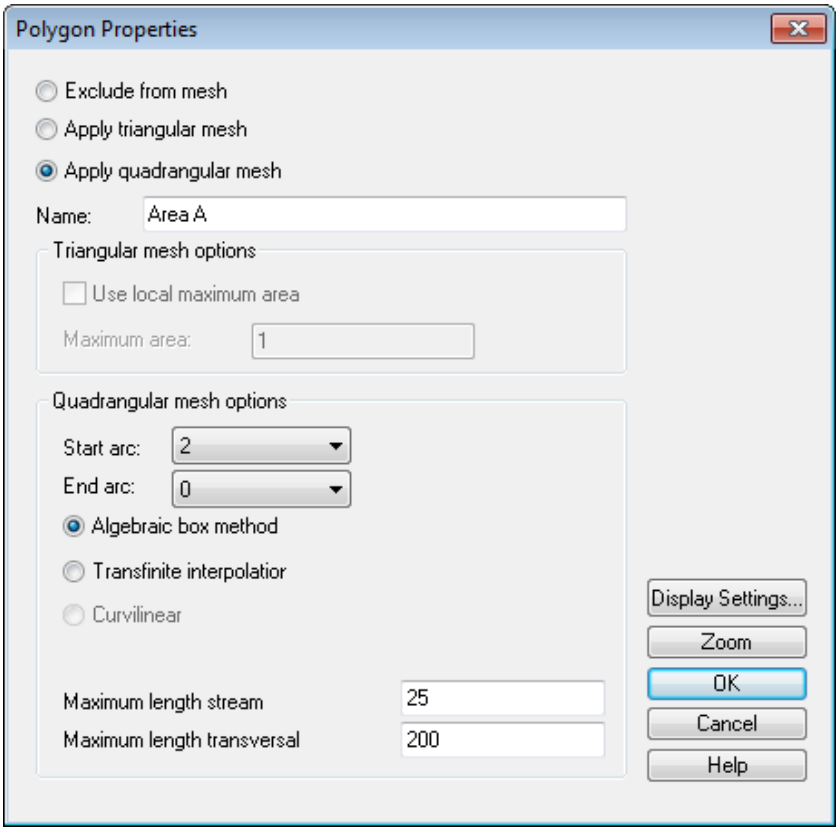


Figure 7.12 Polygon properties for Area A

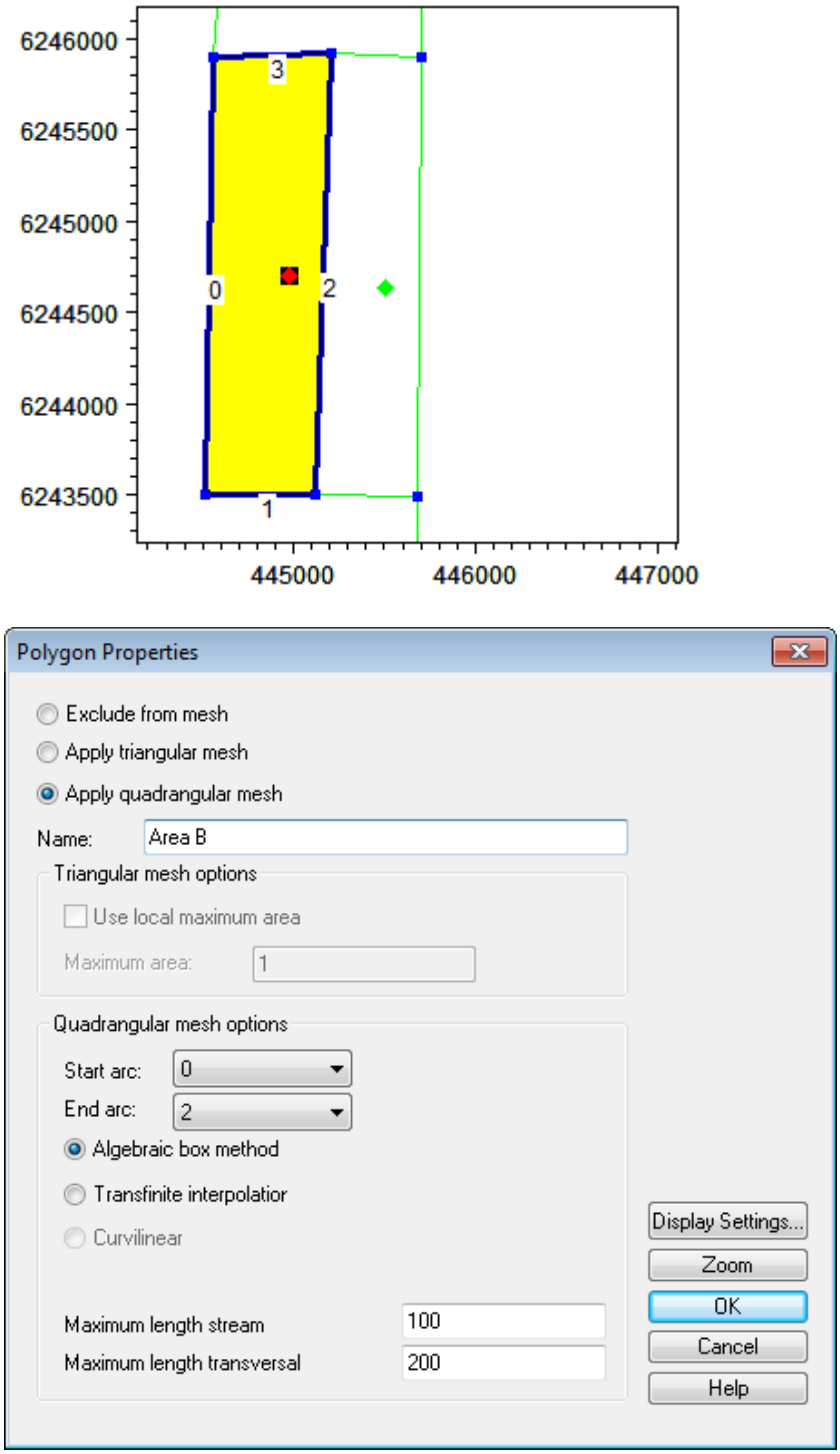


Figure 7.13 Polygon properties for Area B

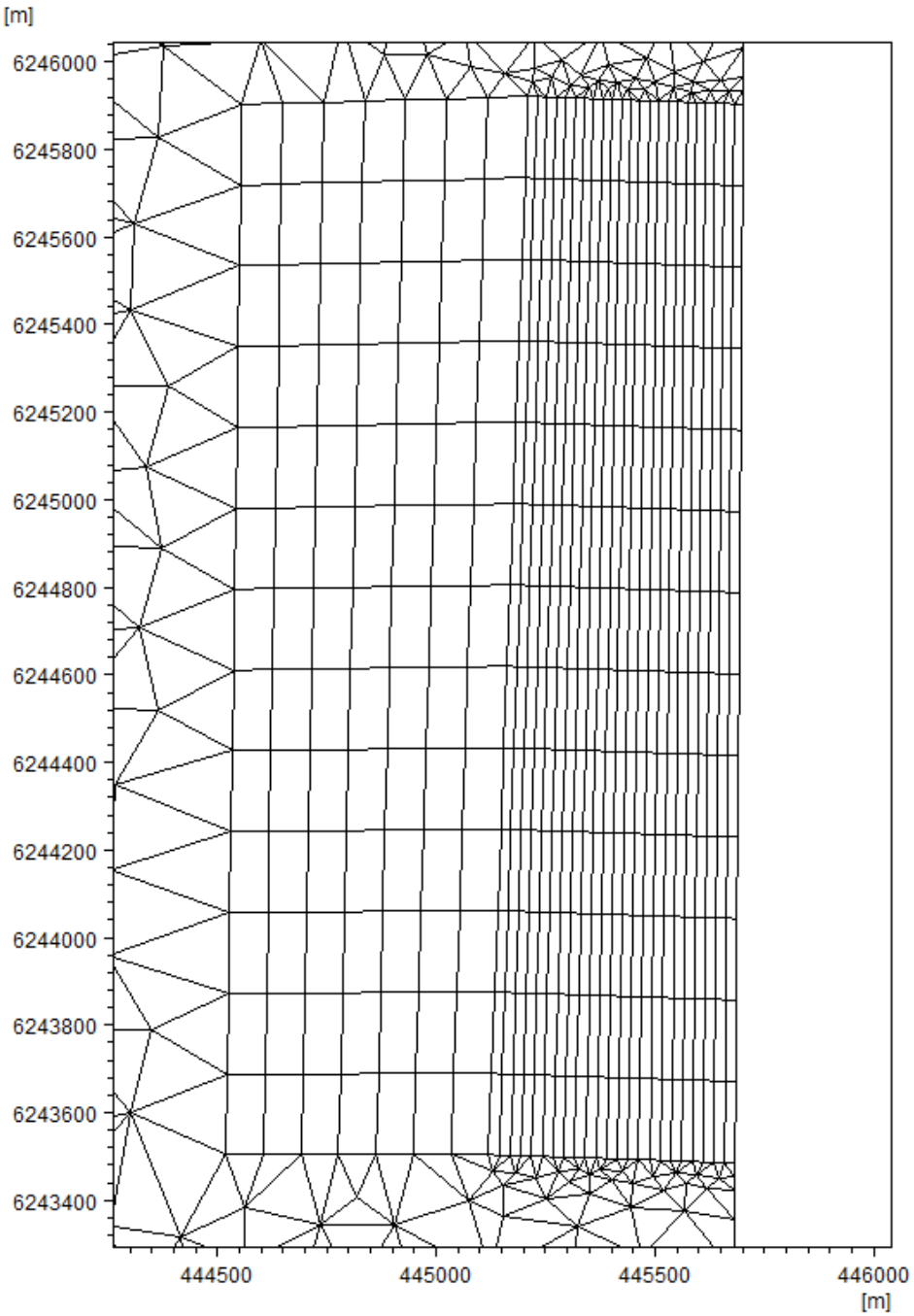


Figure 7.14 Resulting mesh in Area A and Area B

7.2.2 Complex combination of quadrangular areas

The next step is to define polygon properties for Area C. If this is done using the entire area, the mesh generation process will halt issuing an error message.

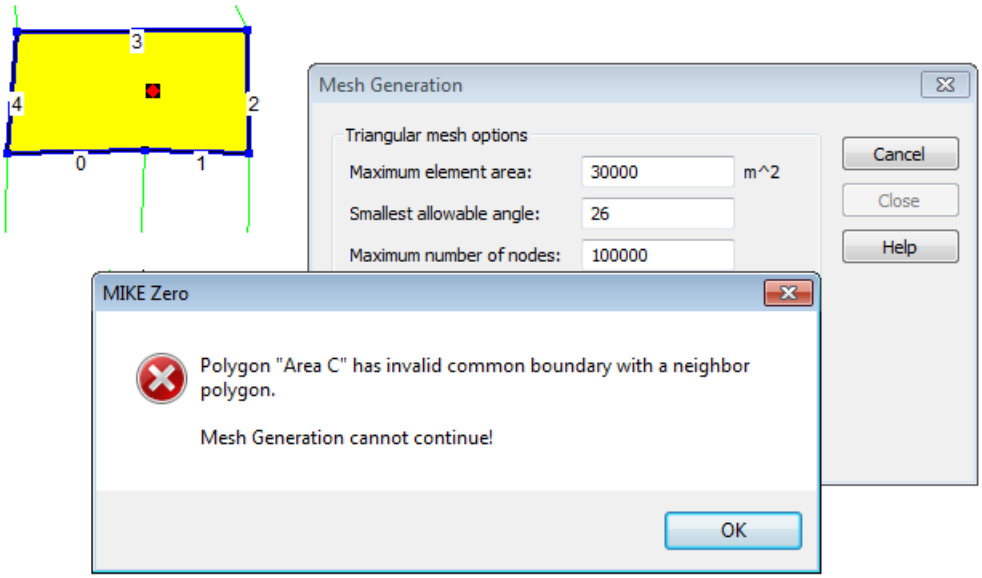
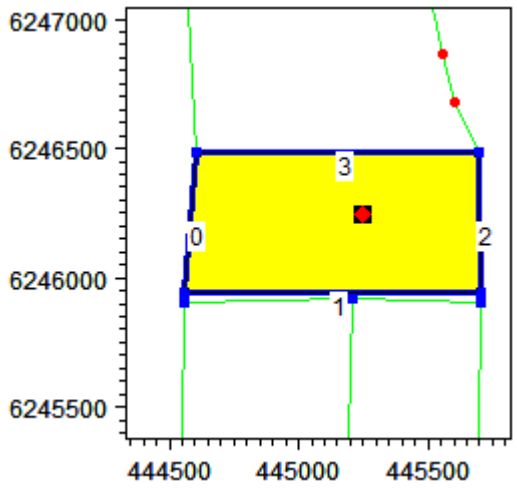


Figure 7.15 Mesh generation result for illegal definition of polygon properties

To overcome this it is necessary to define additional node points such that Area C is separated from Area A and Area B with a (narrow) section of triangular elements, see Figure 7.17.

Please note that the section with the triangular elements performs as a transition area between the quadrangular sections.

In case the Area A, Area B and Area C is to be connected using only quadrangular elements, it is necessary to control the mesh resolution in more detail. This can be done by including vertices on the related boundary arcs. To get an estimate on the number of vertices required, count the number of elements from the mesh in Figure 7.17.



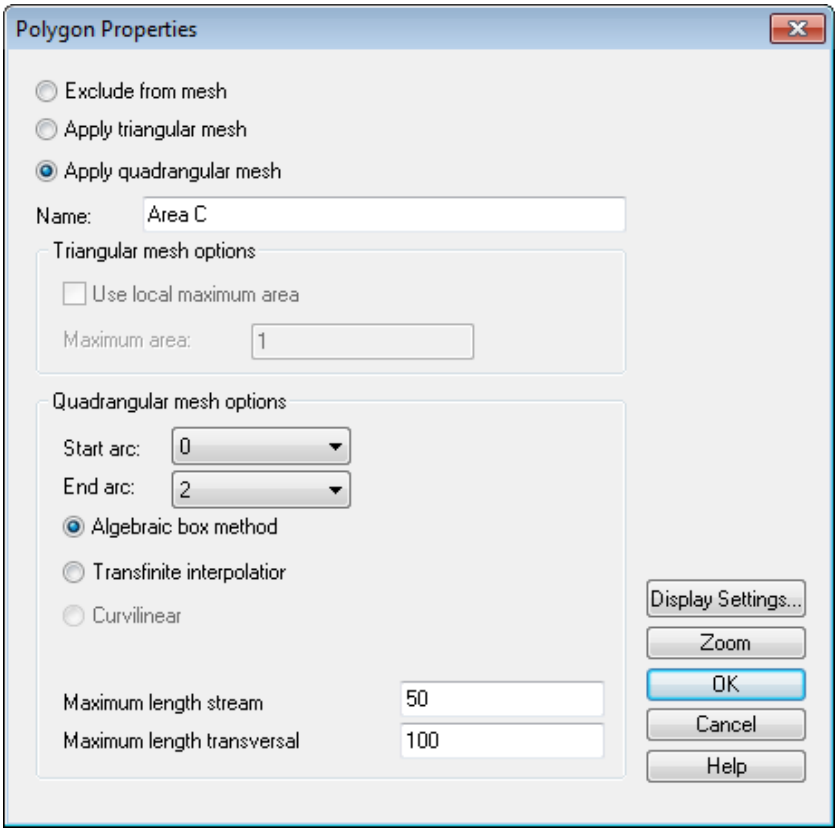


Figure 7.16 Polygon properties for Area C

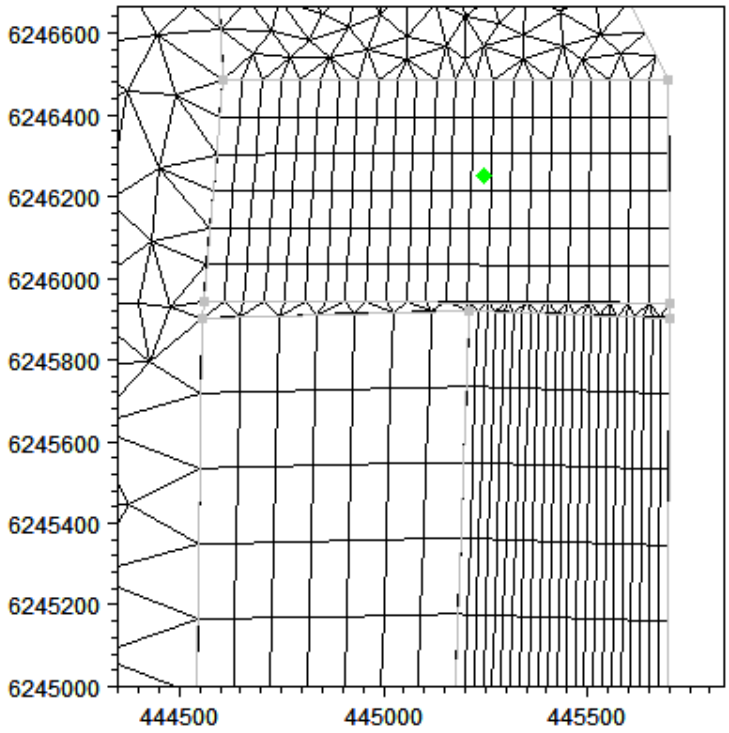


Figure 7.17 Generated mesh using Area A, Area B and Area C.
Small section with triangular elements in transition area

Select the arcs enclosing the transition area and use the Arc Redistribution tool to set the number of vertices to match the conditions from Area A and Area B, cf. Figure 7.18. For more information on Arc Redistribution, please see section 8.1.1 on page78.

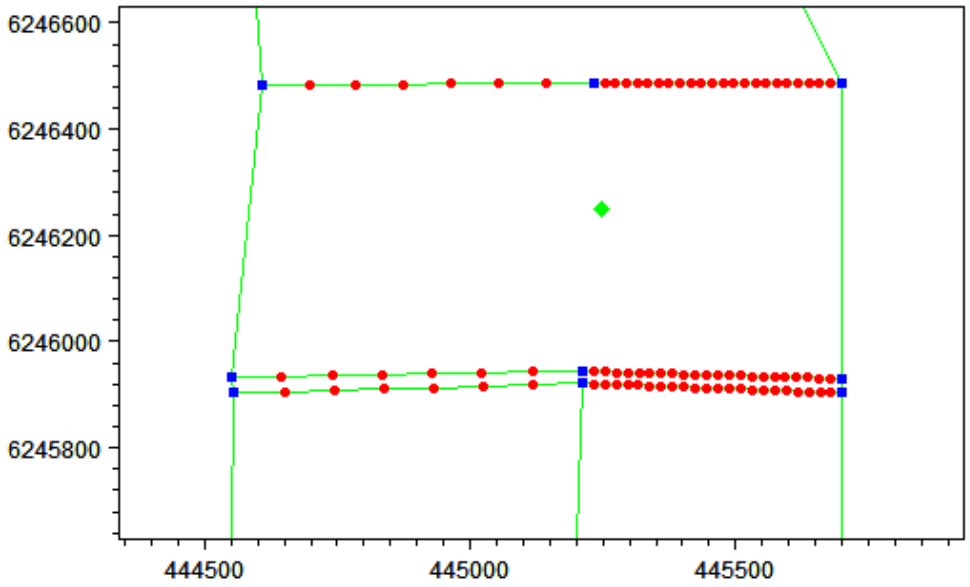


Figure 7.18 Redistribution of vertices along arcs to enable later transition to quadrangular elements

Finally, divide the arc common with area C into two arcs that mirror the conditions from Area A and Area B. This will allow for subsequent editing towards quadrangular elements.

To enforce an equal distribution of elements along the north side of Area C, divide the arc into the same number of segments as on the south side of Area C.

As the mesh element size is restricted along the polygon, define the maximum length as the maximum length of an arc section at the boundary, e.g. 200 m, see Figure 7.19.

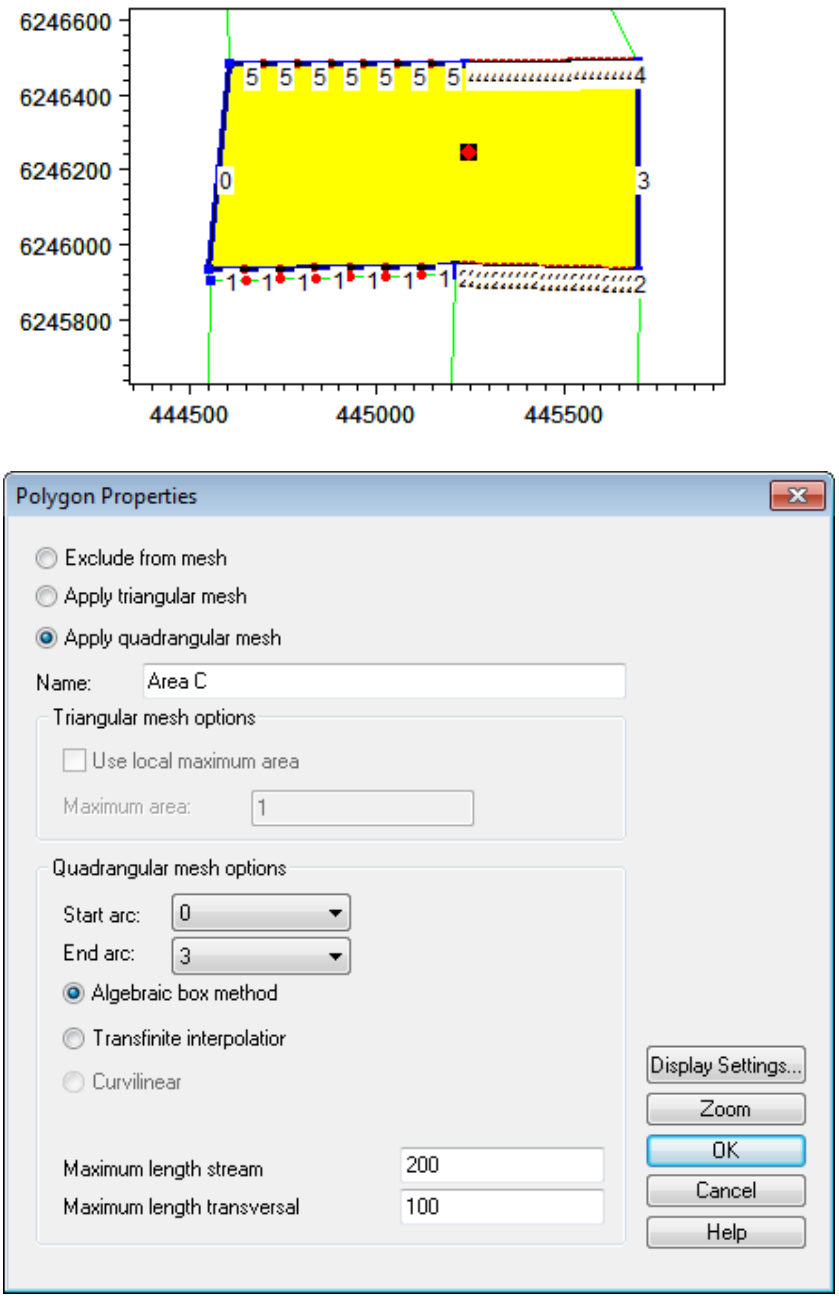


Figure 7.19 Polygon properties for Area C, additional vertices to restrict element size

The generated mesh is shown in Figure 7.20. Having defined the transition area to be narrow, it is a minor task to merge the triangular elements in the transition area to quadrangular elements as shown in Figure 7.21. The resulting mesh is shown in Figure 7.22.

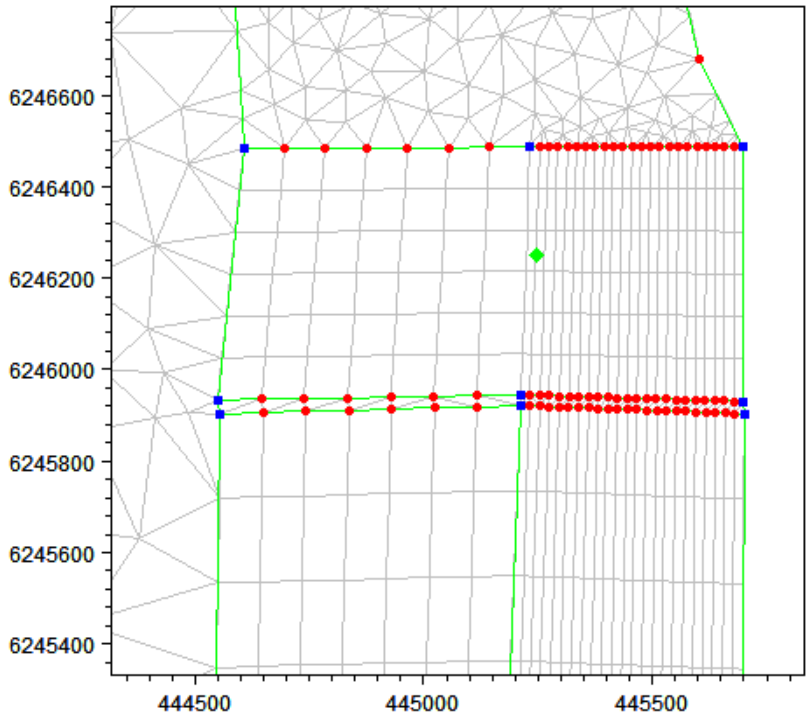


Figure 7.20 Generated mesh using Area A, Area B and Area C.
Small section with triangular elements in transition area

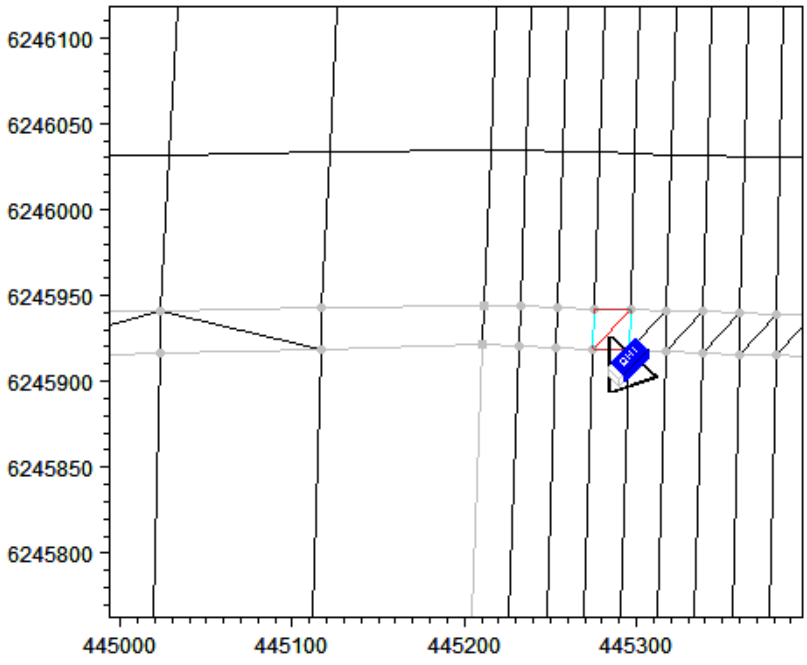


Figure 7.21 Merging two triangular elements to one quadrangular element

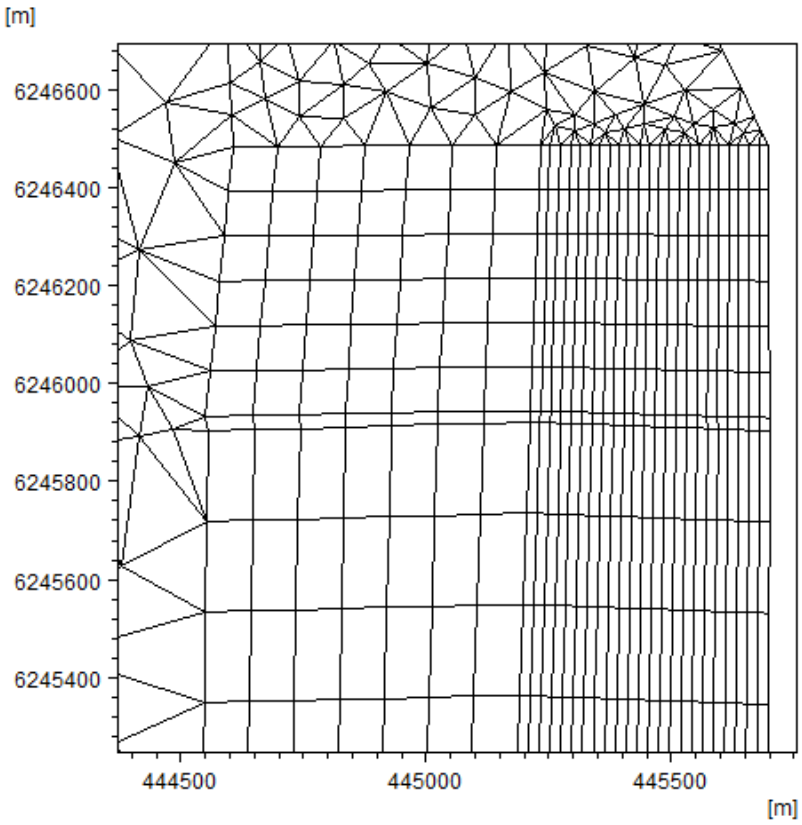


Figure 7.22 Resulting mesh combining Area A, Area B and Area C with purely quadrangular elements

The bathymetry for the resulting mesh can now be obtained by interpolation.

Please note that even if the resulting mesh file is saved in the .mdf file (showing the manual editions), the edition will be lost if you regenerate the mesh.

8 Impact of Tidal Flow

Another type of investigation could be the effect of the tidal water level variation on the water level inside the harbour and associated flushing.

For a first attempt it is assumed that the gates at the entrance to the Thorsminde bay are closed, why the large bay area is not included in the initial mesh.

The coastal outline for this simulation is to be based on the conditions after a harbour extension. The data is extracted from MIKE C-MAP. The xyz files exported from MIKE C-MAP are encrypted, so it is not possible to view the data in a text editor. However, it is possible to import the data into the Mesh Generator.

Please note that all the data exported from MIKE C-MAP is stored in Long/Lat values.

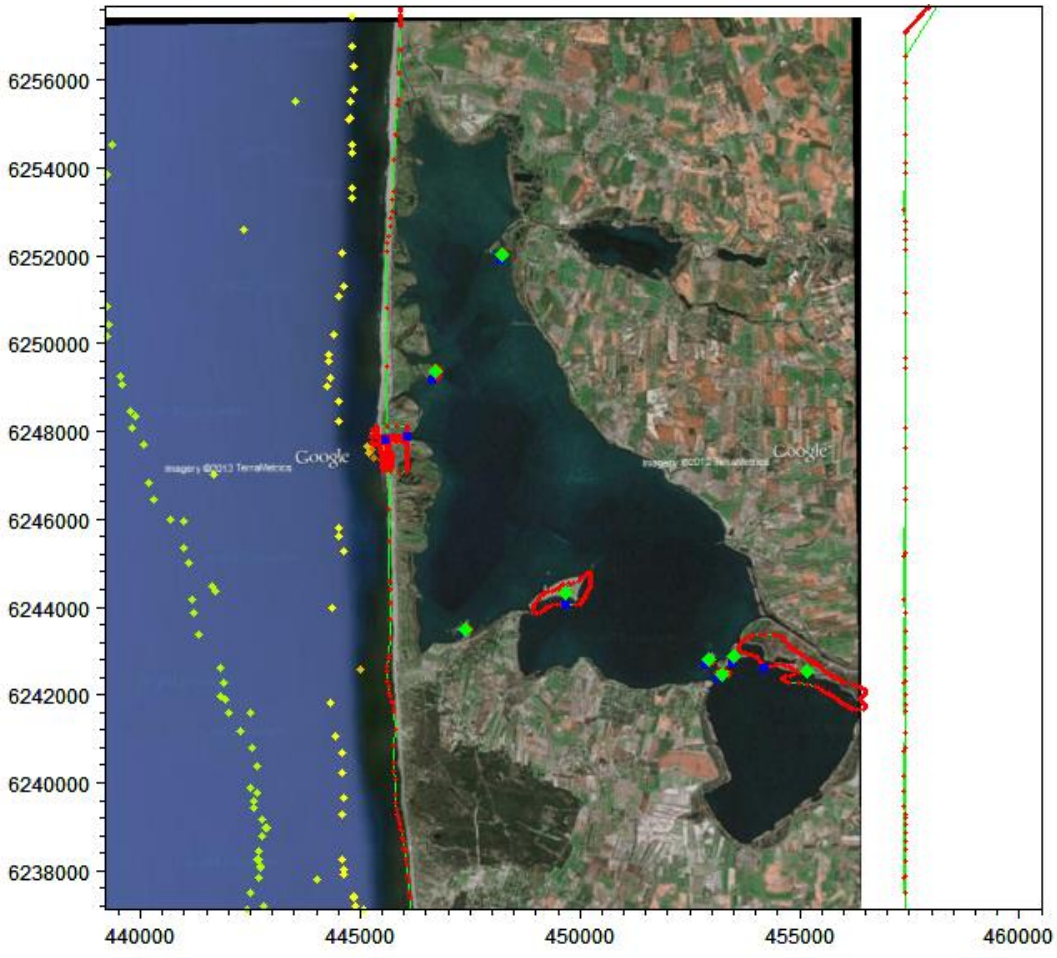


Figure 8.1 C-MAP data displayed in the Mesh Generator. Background image based on Google Earth

8.1 Creating the Land Boundary

For this simulation the focus is on the inner harbour area. The layout of the inner harbour is important as it has a direct impact on the water exchange. The details near the shoreline, however, are not important.

The overall domain area used in the initial mesh generator setup can be used for a template. To include the detailed land contour extracted from MIKE C-MAP, import the land contour in file 'land_CMAP_F.xyz' that contains a detailed land contour for the harbour area.

Please note that all the data exported from MIKE C-MAP is stored in Long/Lat values.

Now modify the arcs to describe one land polyline that defines the shoreline and the inner harbour area.

8.1.1 Redistribute vertices on arc

When digitising land boundaries the number of vertices is often limited. When importing land data from e.g. MIKE C-MAP the opposite may be the case; the number of vertices on the arc lines may be very dense. As all vertices on a boundary arc will result in a mesh node, this may result in very small elements in the final mesh.

So far the boundary modifications in this step-by-step guide have mostly been carried out by manually deleting and editing the vertices and node points. However, it is possible to modify and re-arrange all the vertices along an arc automatically, such that the resolution corresponds with the accuracy needed for the mesh generation. Here, it is an advantage to use the Redistribute Vertices option (cf. Figure 8.2 and Figure 8.3), which makes it possible to preview the outcome of the changes (cf. Figure 8.4) before applying it to the mesh setup, as opposed to the method of simply deleting the excess points. Since different sections of the boundary line may need to be modified in different ways, it may be feasible to divide the boundary line in separate arc sections by redefining a number of vertices to node points.

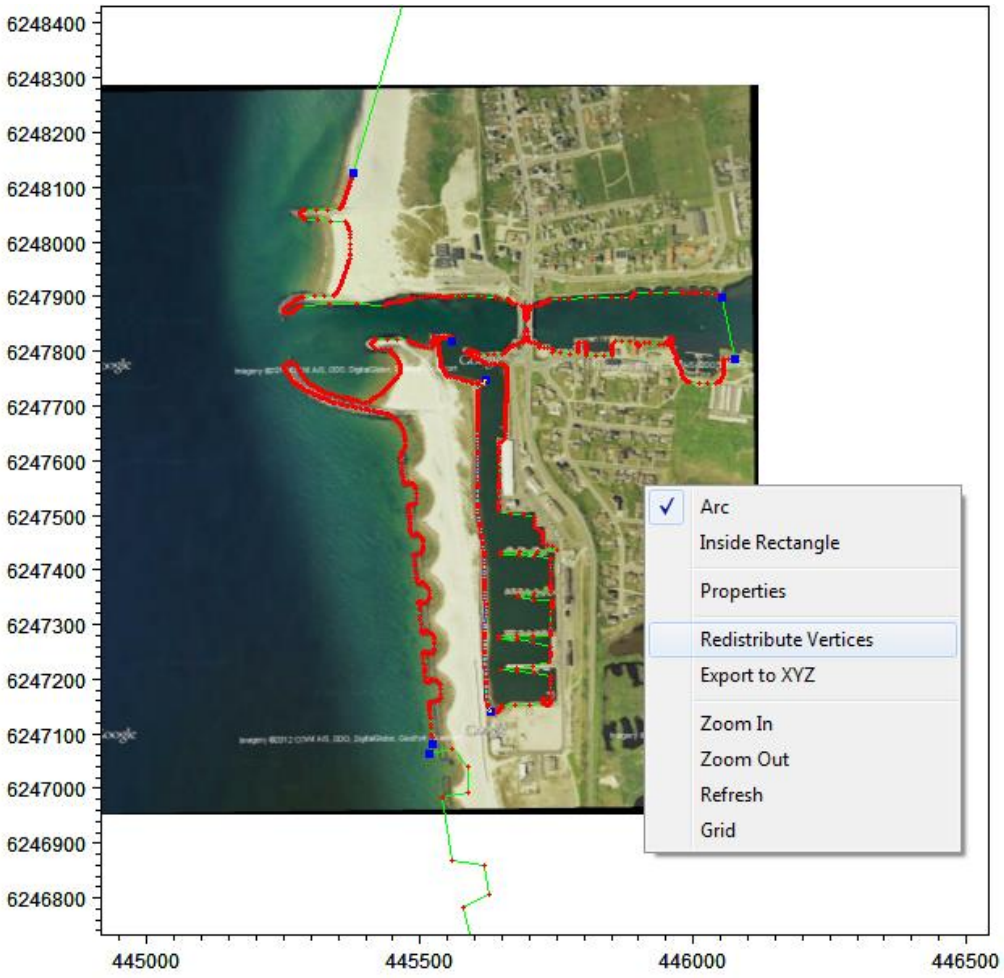


Figure 8.2 Selection of arc in order to redistribute vertices

For long linear stretches it may be best to define the distance between the vertices. With the help of the shown arc properties (cf. Figure 8.3), a constant distance between the vertices may be suggested and only the nearest vertex will be included in the new arc.

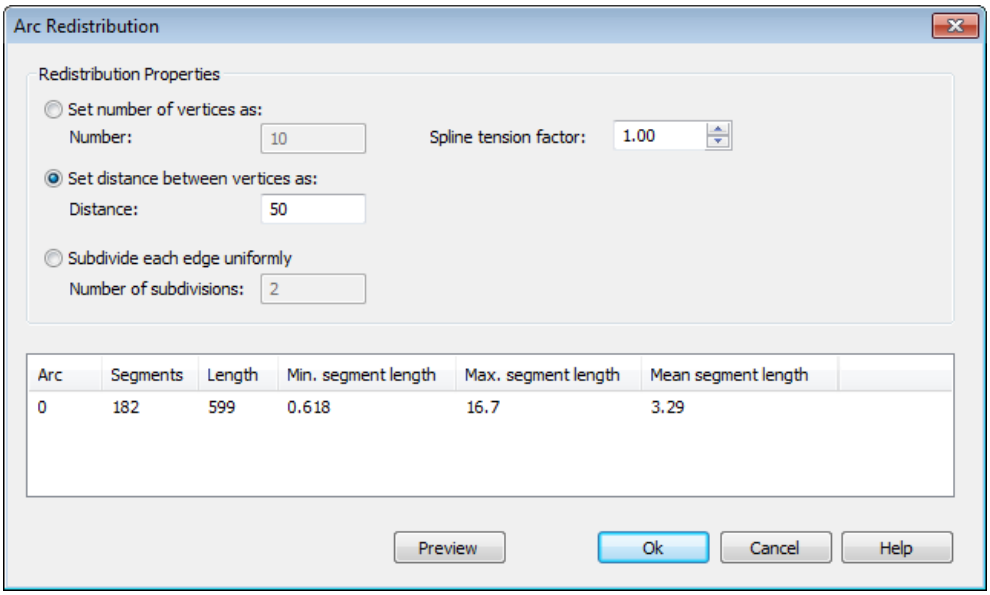


Figure 8.3 Arc redistribution. The initial properties of the selected arc is shown.

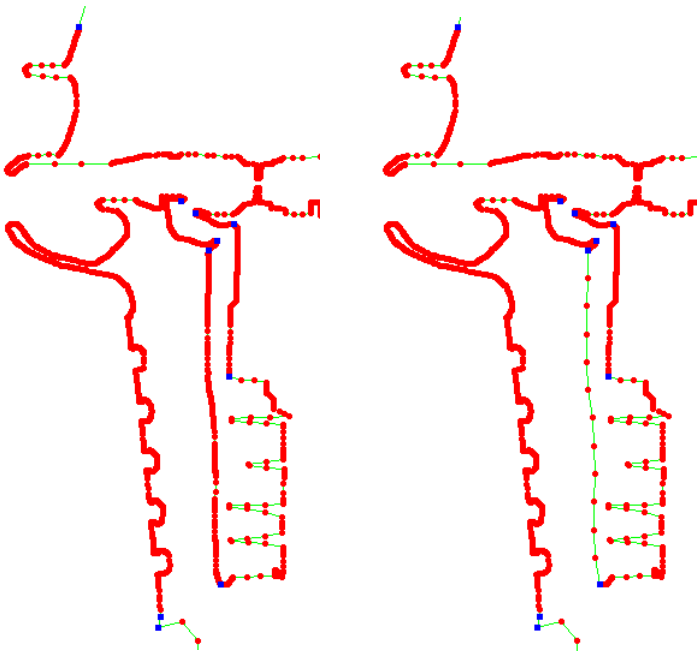


Figure 8.4 Arc redistribution. Before and after redistribution of vertices along the west side of the inner harbour.

For arcs that are to be smoothed anyway it may be more feasible to define a fixed number of vertices (cf. Figure 8.5). This way a new representative arc is created based on a spline tension factor. Note that the vertices for the new arc are not necessarily located on the existing arc. It is possible to preview the new resulting arc before pressing OK to evaluate if the redistribution properties gives the required result.

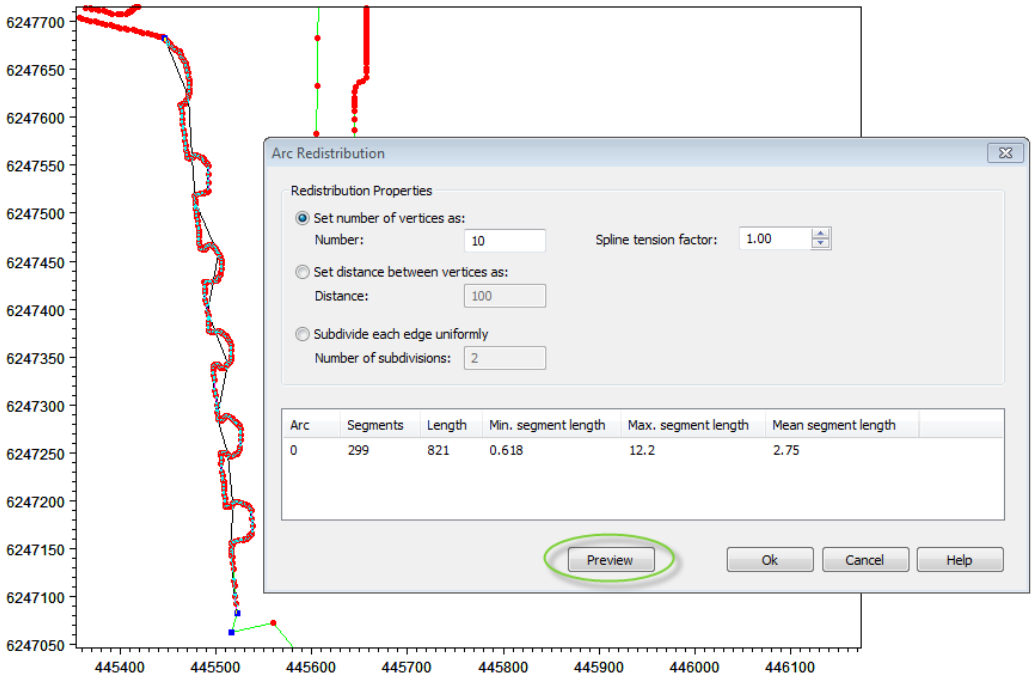


Figure 8.5 Arc redistribution. Preview of new arc at coastline using fixed number of vertices.

Usually the process of modifying a mesh boundary will consist of the combined use of all the different ways of modifying arcs and points.

Note that the processes of modifying the boundary data may take some time. This time, however, is saved later as the mesh will perform better in the simulation. It is a good idea to save the mesh setup in different stages of the editing and in separate files, in order to have the option to revert to an earlier stage of the editing.

The land boundary may look as shown in Figure 8.6 after some editing:

Please note that the land boundary will probably have to be edited even more after the initial mesh generation as the analysis tool will point out elements that prevent an optimal solution.

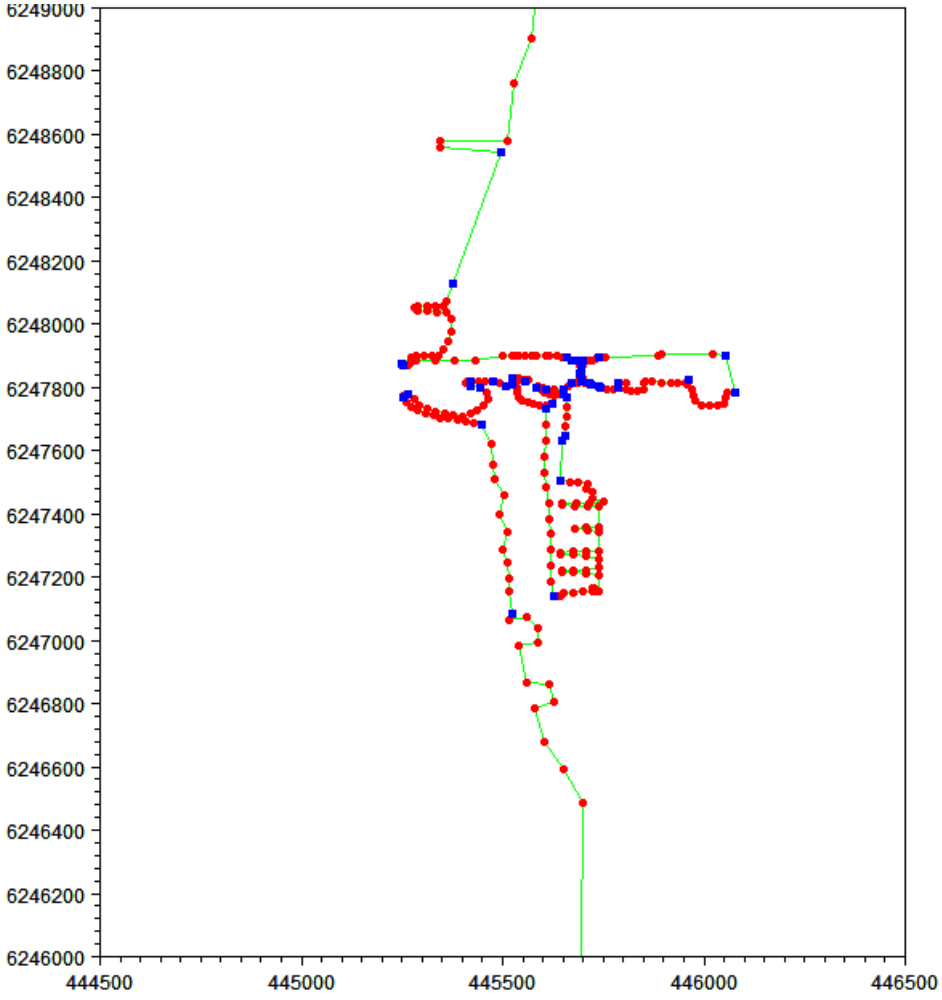


Figure 8.6 Land boundary after editing.

8.2 Specification of Domain

By assuming a constant water level variation in front of the harbour entrance due to tide, the model area outside the harbour can be limited. Thus, the area in front of the harbour entrance is limited by one boundary line, onto which a constant time-varying water level is to be defined in the simulation.

In order to gain details of the flow field inside the harbour an extra arc is defined in order to define two polygons with individual mesh element settings, cf. Figure 8.7 and Figure 8.8.

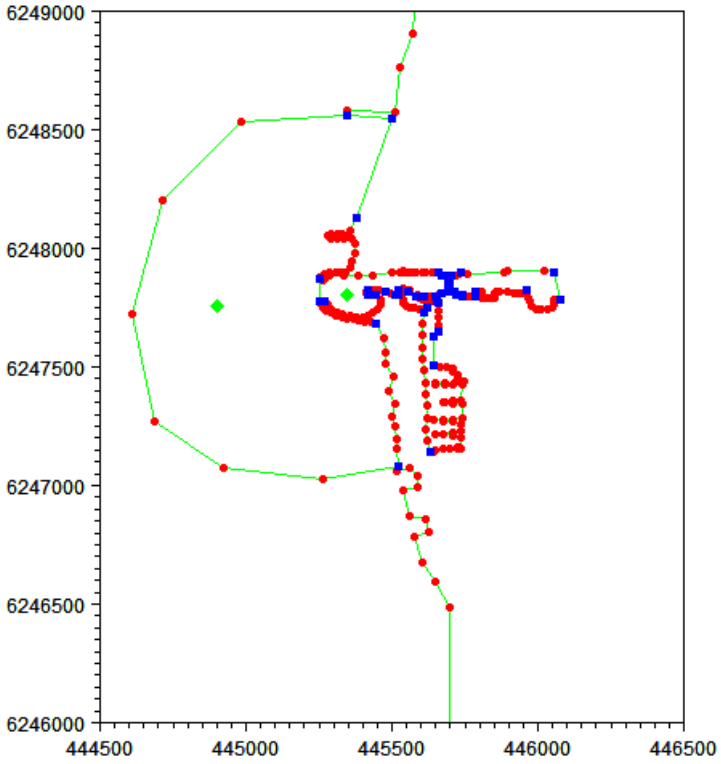


Figure 8.7 Domain for simulation. Two polygons defines inner and outer area.

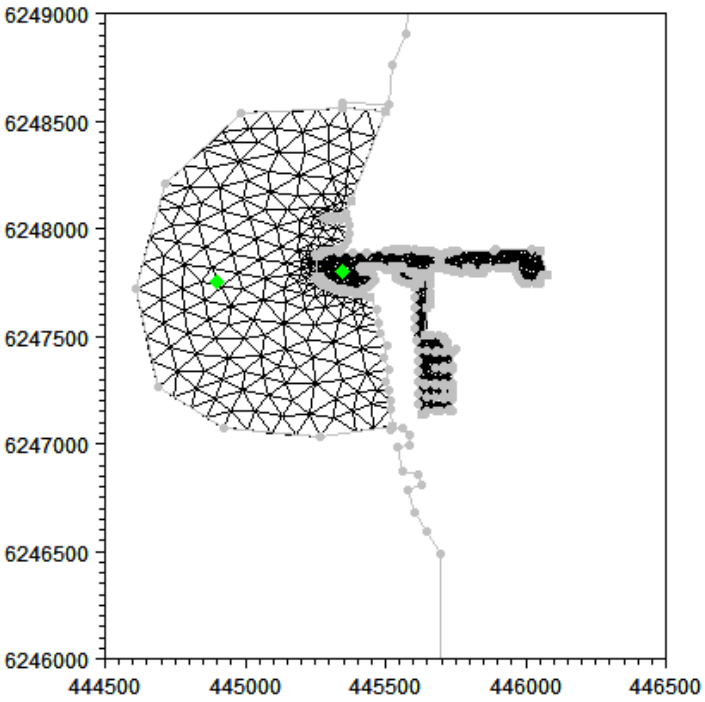


Figure 8.8 Generated mesh. The detailed mesh in the inner harbour (2784 elements)

8.3 Using Break Lines

When applying the scatter data it may occur that some of the scatter data should be excluded from the interpolation, even if the scatter data is in adjacent areas. This is the case of the inner harbour where the bathymetry data defining the coastal area does not affect the depth of the harbour. In cases where the scatter data is divided into separate files, e.g. one for the coastal area and one for the local harbour area, it is possible to utilise the scatter data prioritisation utility to explicitly define which scatter data that is to be applied in the interpolation.

However, if all the scatter data is included in the same file, the interpolation will try to use all the scatter data within the surroundings, even if the area is not connected. In this case break lines can be applied when interpolating so that the water depth in the harbour becomes independent of the water depth defining the beach.

For this case assume that the depth in the harbour is constantly dredged to a minimum depth of 5 m. To include this in the Mesh Generator setup, create a new scatter data file containing the depth at a number of locations within the harbour. In addition, add the scatter data for the local survey lines to describe the water depths outside the harbour, cf. Figure 8.9.

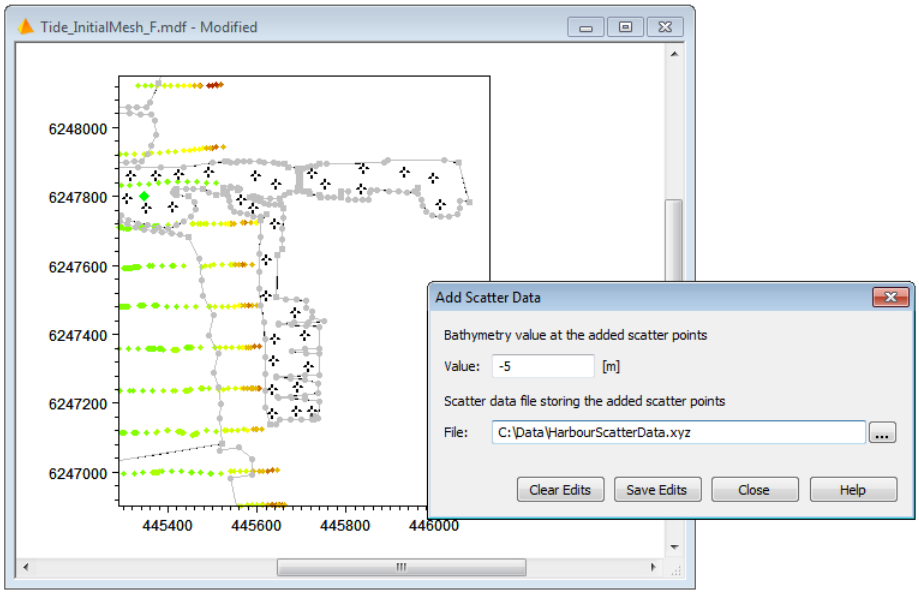


Figure 8.9 Adding scatter data to new file

When just interpolating the scatter data it is seen that the scatter data from the local lines from the sea will affect the water depths inside the harbour, see Figure 8.10. The effect can be minimised by reducing the extrapolation. This, however, may affect other areas of the domain where a large extrapolation is needed.

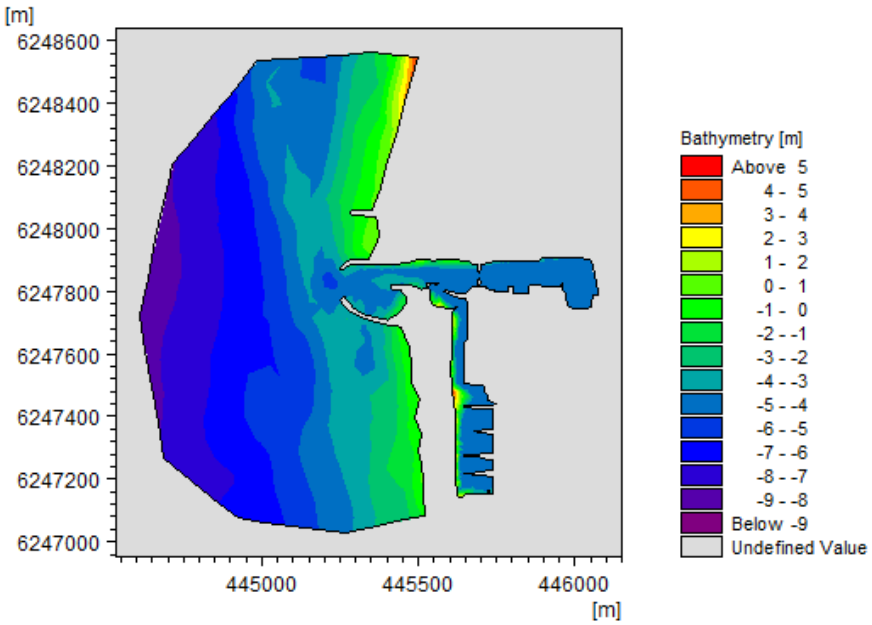


Figure 8.10 Resulting bathymetry with standard interpolation

The solution is to create a break line that prevents the use of the un-wanted scatter data points in the interpolation of the area. A break line is defined using an icon in the Boundary Definition toolbar:

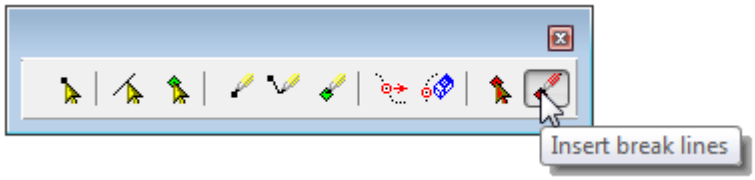


Figure 8.11 Inserting a break line from the Boundary Definition toolbar.
Note that you have to be in arc editing mode for the toolbar to activate

Define a line that divides the scatter data by the sea and the scatter data to be used in the harbour area as shown in Figure 8.12. The interpolation will then only consider the scatter data within the harbour area when calculating the depths in here.

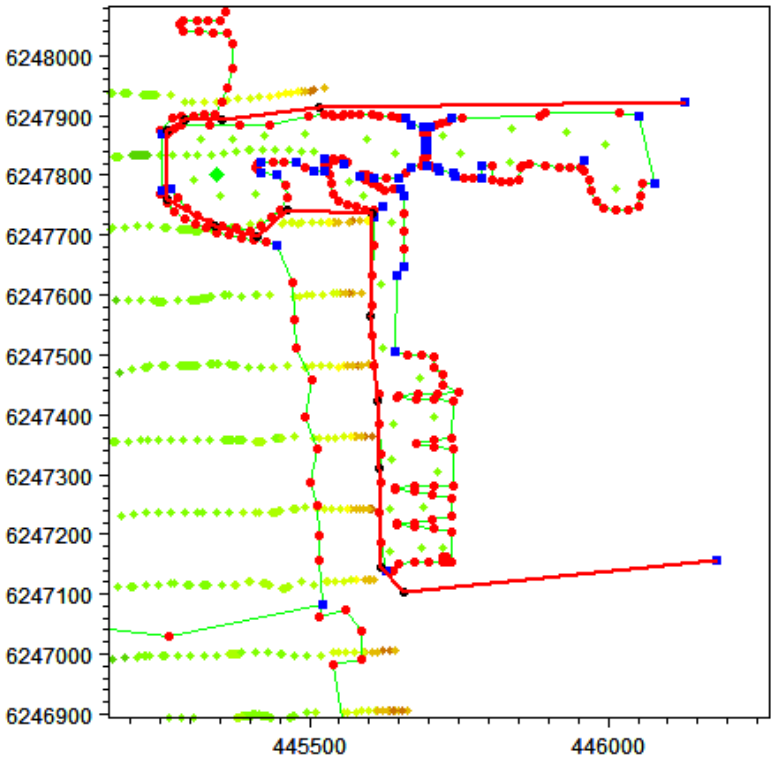


Figure 8.12 Definition of break line

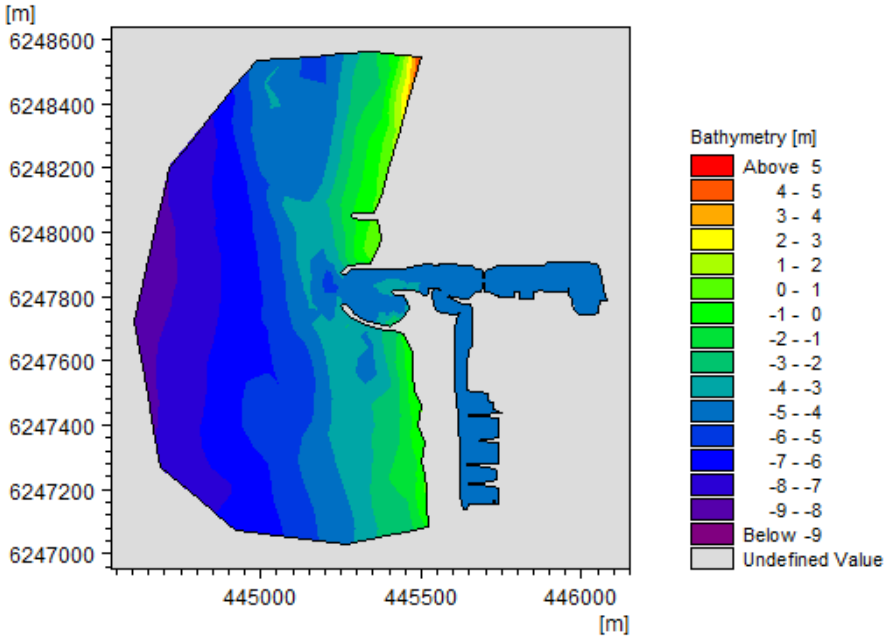


Figure 8.13 Resulting bathymetry using break line

This bathymetry can now be used for simulating the flushing of the harbour during tide. Please see Section 8.5 for results.

8.4 Increasing the Model Domain

The flow field through the harbour entrance for tidal simulations may change if the water volume within the Thorsminde bay is added to the simulation.

For this particular simulation, the accuracy of the layout of the inner bay is of less importance, as long as the overall area of the inner bay is correct.

The previous mesh generator setup can be used as a template.

Add the image of Jylland (GE_Regional.png) to the background and digitise the bay area by creating a new arc. Attach the new arc to the existing setup and define the bay as a separate polygon, cf. Figure 8.14.

After the mesh generation and interpolation the resulting mesh may look like the one shown in Figure 8.15.

Figure 8.14 Adding inner bay to mesh setup. Background image based on Google Earth

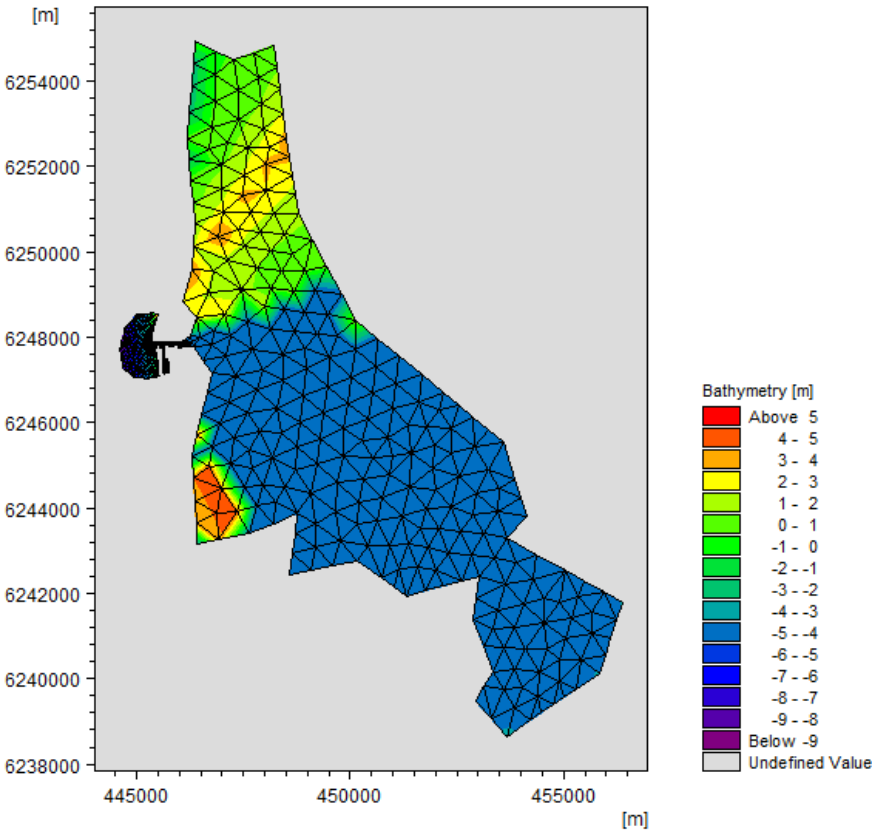


Figure 8.15 Interpolated mesh including the inner bay. Bathymetry values are not edited

Please note that the bathymetry values in the inner bay near the outlet are based on the scatter data in the harbour because of the break lines. However, the main part of the bay is also influenced by the scatter data in the sea.

To overcome this, one can add more break lines to be included during the interpolation. Sometimes, however, it is more convenient to modify the bathymetry data after saving the mesh to a mesh file.

Open the saved mesh file in the Data Viewer and select the elements within the bay area. Then set the overall bathymetry value to -5m, similar to the value in the harbour area, cf. Figure 8.16.

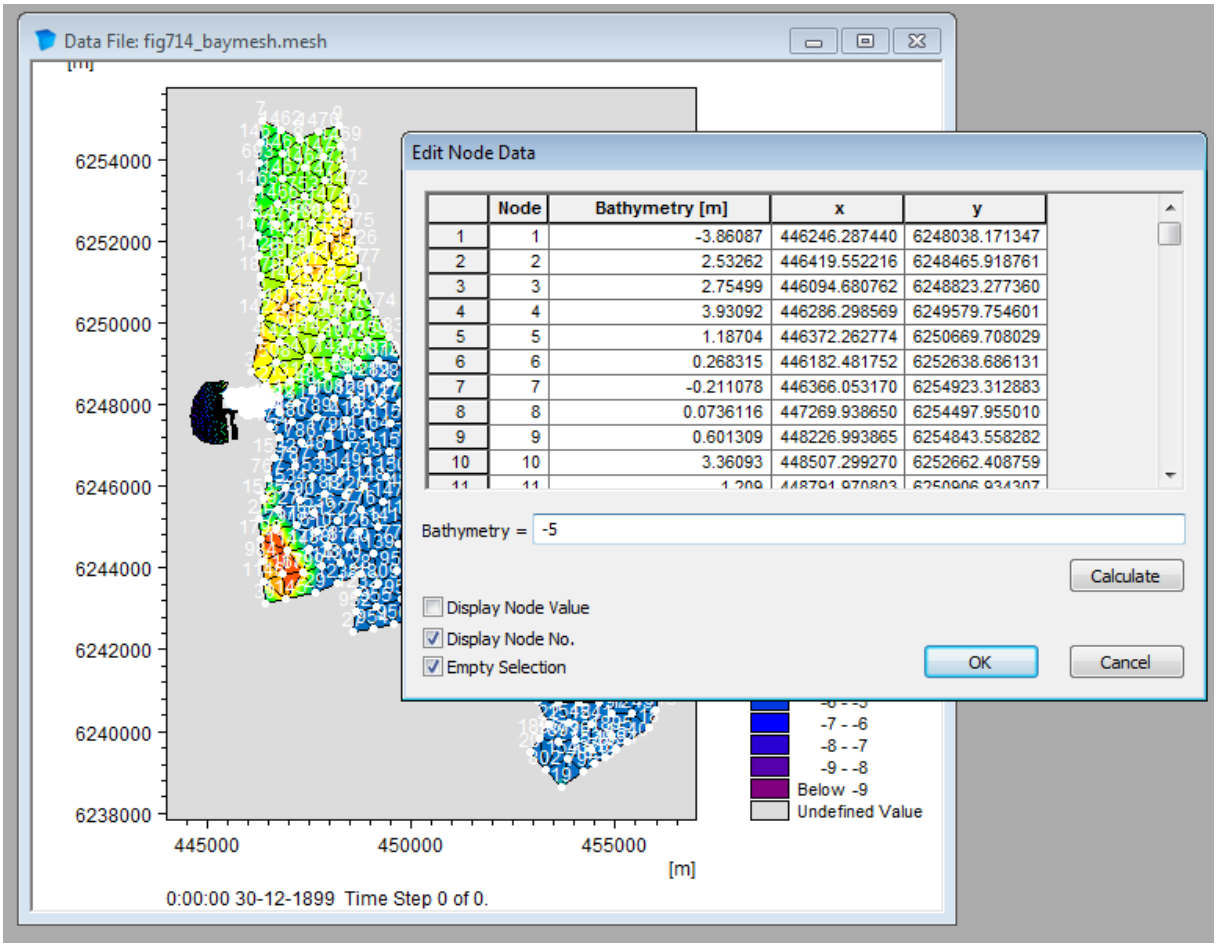


Figure 8.16 Modification of bathymetry values using the Data Viewer

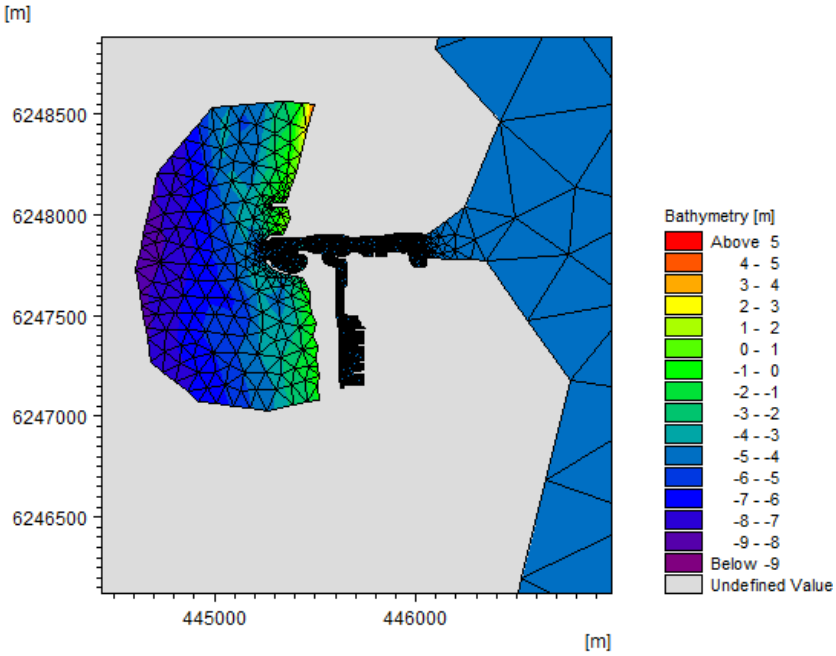


Figure 8.17 Detail of resulting mesh bathymetry after modification

8.5 Simulation Results

To evaluate the effect that the tide and the additional bay area have on the flow in the harbour area, different model setups are simulated. Some results are shown in Figure 8.18 - Figure 8.22.

Note that the flow field is very dependent on whether the gate is open or not.

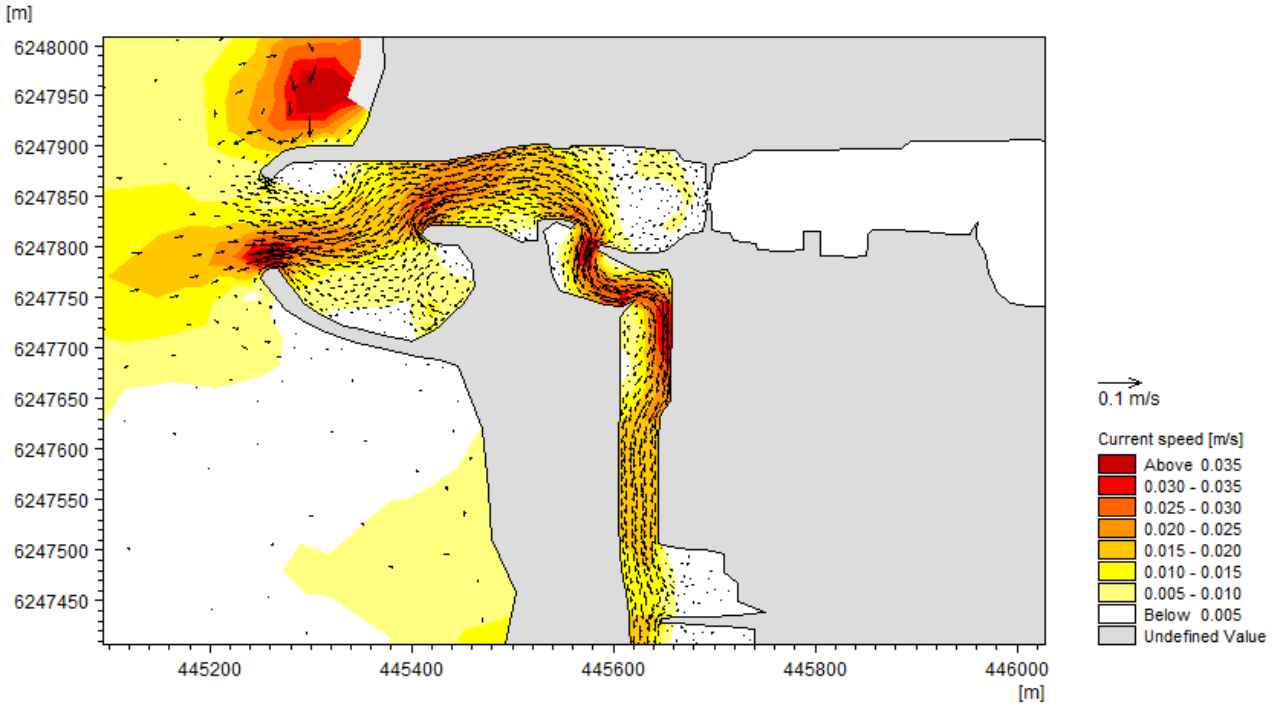


Figure 8.18 Detail of resulting flow field in harbour when gates are closed

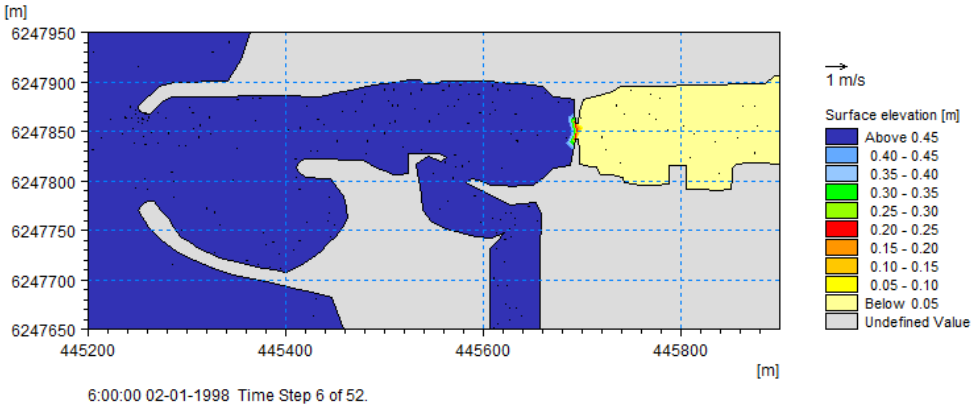


Figure 8.19 Detail of resulting flow field and surface elevation in harbour when gates are closed at high tide

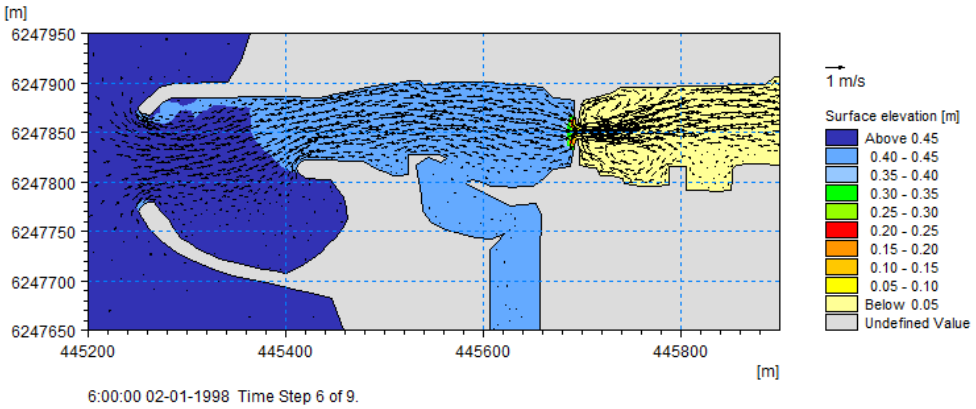


Figure 8.20 Detail of resulting flow field and surface elevation when gates are open at high tide

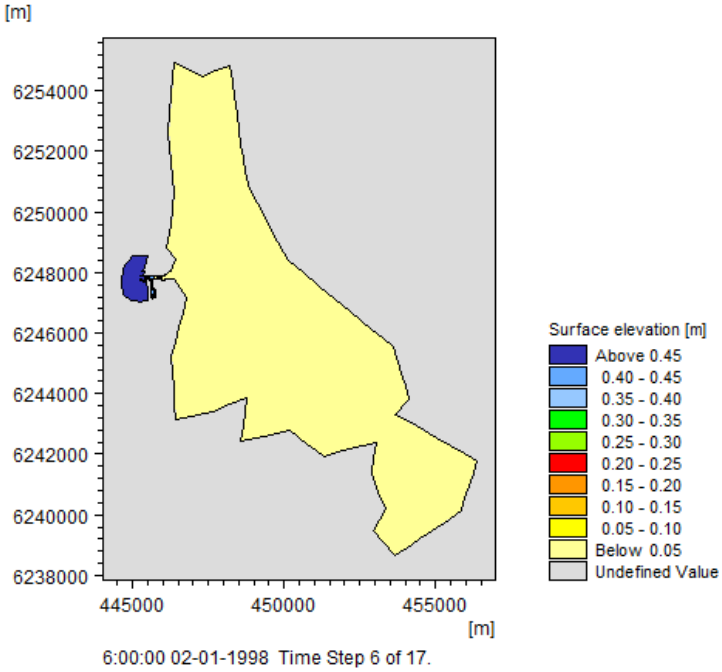


Figure 8.21 Resulting surface elevation in the domain when gates are open at high tide

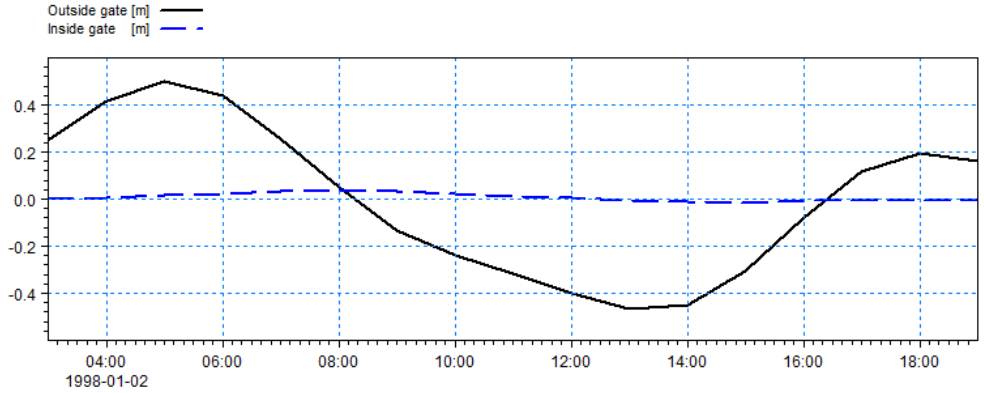


Figure 8.22 Extract of surface elevation 100 m on each side of the gate

9 Phase II of Previous Investigation

It happens that model setups used in older investigations may be needed again because the boundary conditions have changed, e.g. a new harbour layout is being made. This will often cause an investigation as to how the changed layout will affect the flow pattern. Preferably this investigation would be carried out before any constructions take place, but sometimes the investigations are carried out because something un-expected has happened.

In some cases the original .mdf file has been saved. Then the modifications to a new layout are fairly simple, - just modify the settings with the new information and re-generate the mesh.

In case it is only the bathymetry data that has changed, e.g. due to a new harbour survey, it is sufficient to import the mesh directly into a new .mdf file, add the new scatter data and then re-interpolate the bathymetry using the new values.

However, sometimes the only available data is the existing mesh file and the information about the new land boundary and possible changes in the bathymetry. In that case, the generation of a new mesh bathymetry requires more work. Such a case is described in the following using two different methods: modifying the mesh directly or re-establishing the mdf file based on the old mesh information and use this as a starting point.

Consider the case with the existing mesh bathymetry as displayed in Figure 9.1.

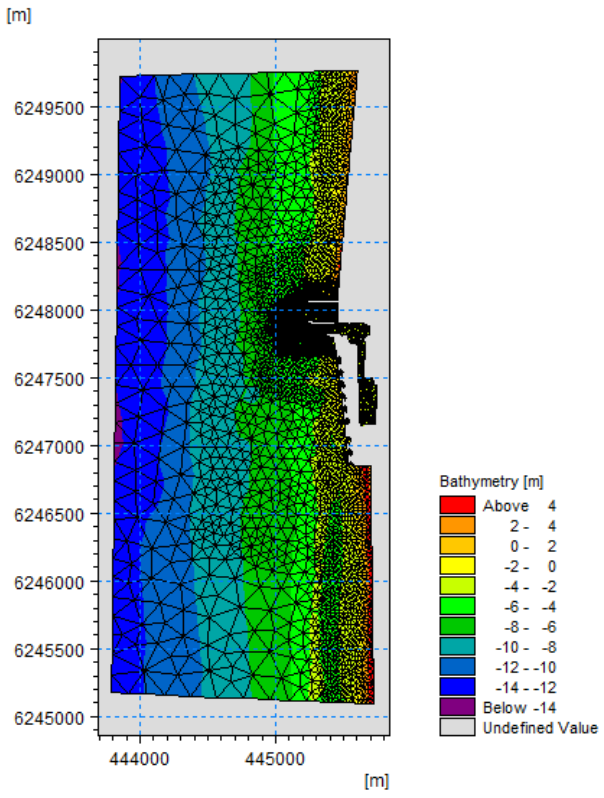


Figure 9.1 Existing mesh bathymetry

The bathymetry contains 4211 nodes and 7876 elements, with smaller elements defining the area around the harbour entrance.

After some years a new layout of the harbour entrance has been defined. Now it is necessary to update the existing mesh with the new information to simulate the changes caused by the new layout.

The information regarding the new harbour layout is derived from extracting land boundary data from MIKE C-MAP (land_CMAP_F.xyz). The outline of the changes needed is indicated in Figure 9.2. This is similar to the data displayed in Figure 8.1.

Obviously the bathymetry values will have changed after the construction, but this is not considered in the present exercise.

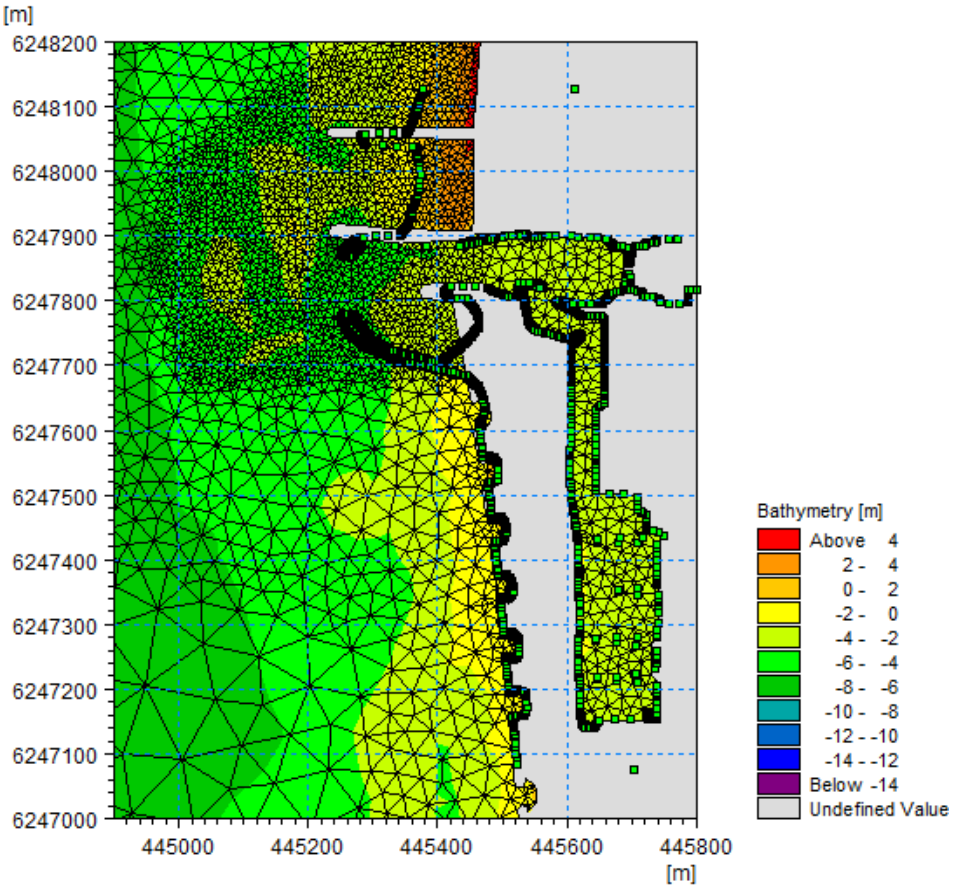


Figure 9.2 Existing mesh bathymetry.
Outline of new land boundary is indicated by xyz point display

9.1 Modifying Existing Mesh File Directly

If the number of changes to the mesh are minor, it may be most practical to just modify the existing mesh directly by using the Mesh Editing facilities, manually.

Create a new mesh generator setup by loading in the existing mesh directly using the Import mesh command, and say **No** to generate information for Arc Editing. Keep the editing mode as Mesh Editing. Next import the Land boundary as scatter data as shown in Figure 9.4. The scatter data will be used as a guide line for the modifications.

The task ahead is now to modify the mesh to match the new layout and then interpolate the bathymetry using the bathymetry values from the existing mesh.

The Mesh Editing toolbar shown in Figure 9.3 contains a number of different tools that can be used for this purpose.

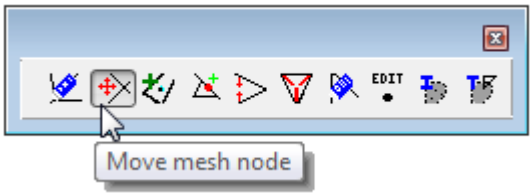


Figure 9.3 Mesh Editing toolbar

9.1.1 Moving mesh nodes

Try to move the node points to a position such that the outline of the new structures is shown, cf. Figure 9.4.

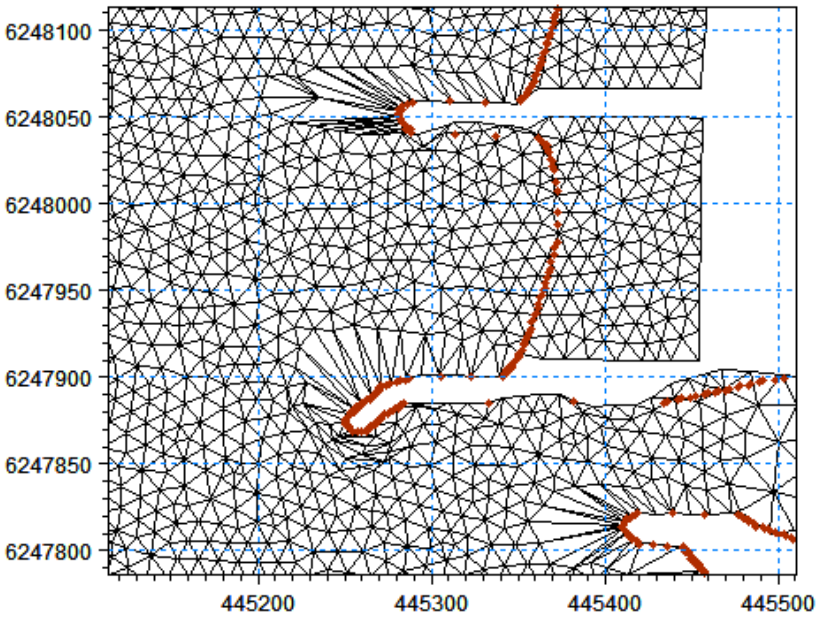


Figure 9.4 Moving mesh nodes in existing mesh bathymetry. Land scatter data shown as points

Note that this may cause some of the areas or the angles in some of the triangles to become very small. In order to analyse the changes use the Analyse tool to identify areas where the mesh modification could be improved, cf. Figure 9.5.

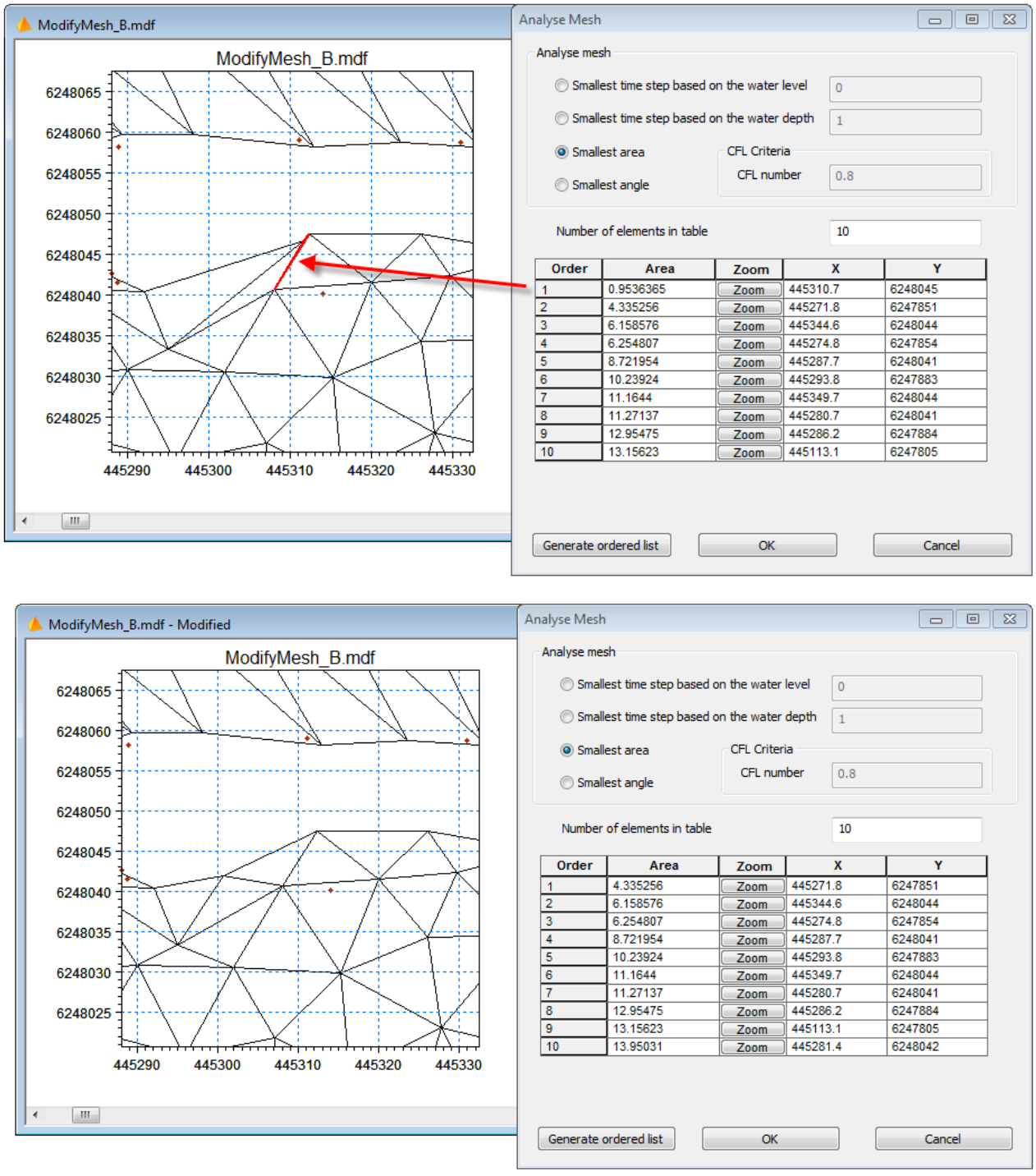


Figure 9.5 Example of analysis of modified mesh. Upper: Analysis result before editing problematic mesh element. Lower: Analysis result after editing

However, it may be easier to re-triangulate the affected area.

9.1.2 Re-triangulate selected region

Moving mesh nodes will often give rise to mesh elements that are small or have small angles. In order to optimise the mesh after the changes it is possible to re-triangulate the mesh elements in a selected area, cf. Figure 9.6. This will leave the mesh nodes outside the selected area untouched.

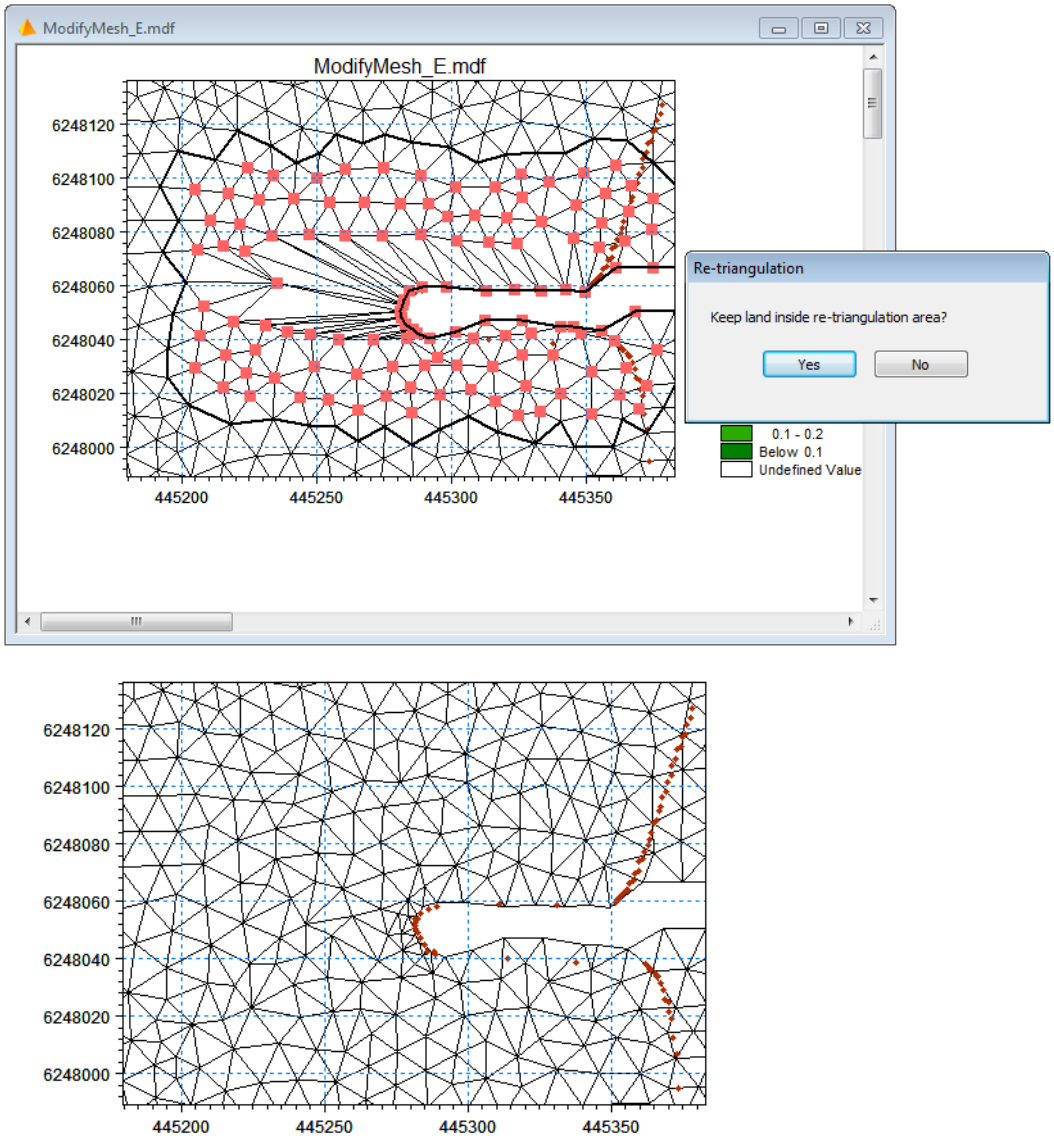


Figure 9.6 Re-triangulation of selected region.
Upper: selected area to re-triangulate
Lower: resulting new mesh after re-triangulation using maximum area of 100

Keep the land inside the triangulation area. This will ensure that the boundary node points will keep the original position.

Note that re-triangulation can be used as a way to increase the number of mesh node points in a certain area before a manual adaption of the mesh begins.

9.1.3 Adding node points

In some areas the moved points may cause single mesh elements to be quite large.

Apart from the option to re-triangulate a selected region, it is also possible to improve the mesh resolution by adding an extra mesh node into the mesh element as shown in Figure 9.7. You have the option to re-triangulate the adjacent elements as well. In most cases this will be feasible as the overall mesh layout will be better, but in some cases it is better to maintain the layout of the adjacent mesh elements, cf. Figure 9.8.

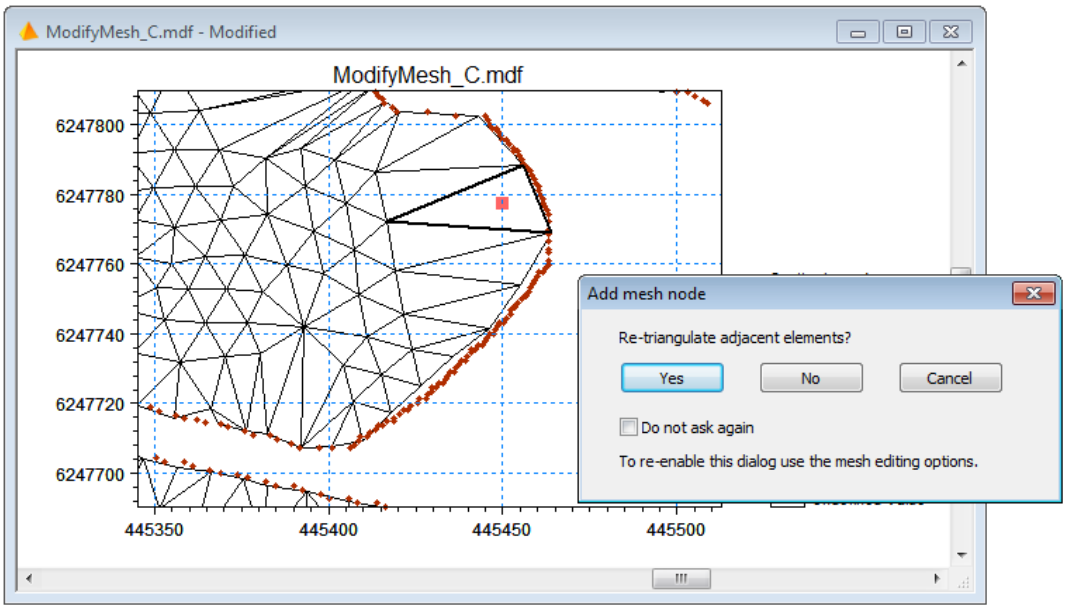


Figure 9.7 Adding a mesh node

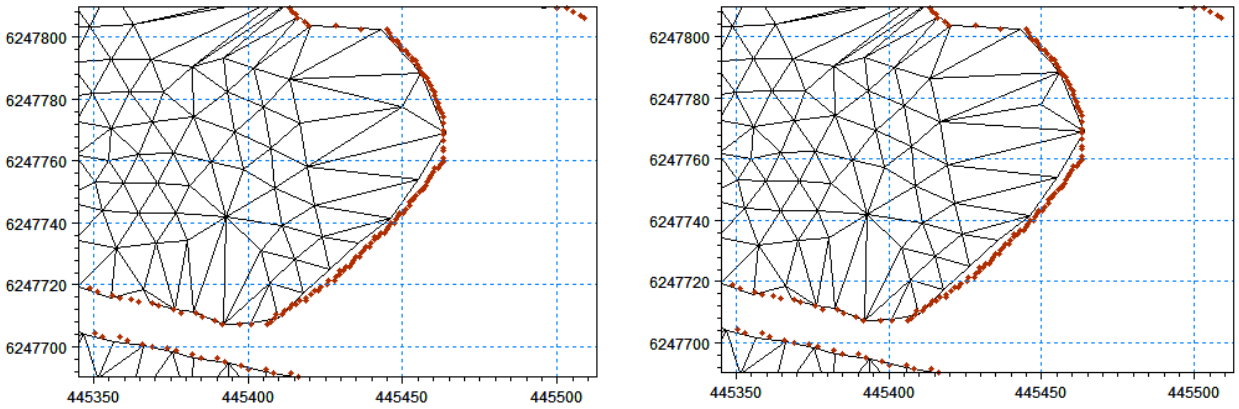


Figure 9.8 Resulting mesh after adding mesh node.
Left: including re-triangulation of adjacent elements, Right: no re-triangulation

Similarly, it is possible to add a node point to a boundary. In this case only the affected element is modified.

9.1.4 Deleting mesh nodes

Try to delete the mesh nodes that are located in areas where new the new structure should be. When asked for method, select ‘Make land’, cf. Figure 9.9. This will reduce the number of mesh elements and it will be easier to move the remaining mesh nodes to a position to outline the new structure. Figure 9.10 shows a resulting mesh after deleting and moving mesh nodes.

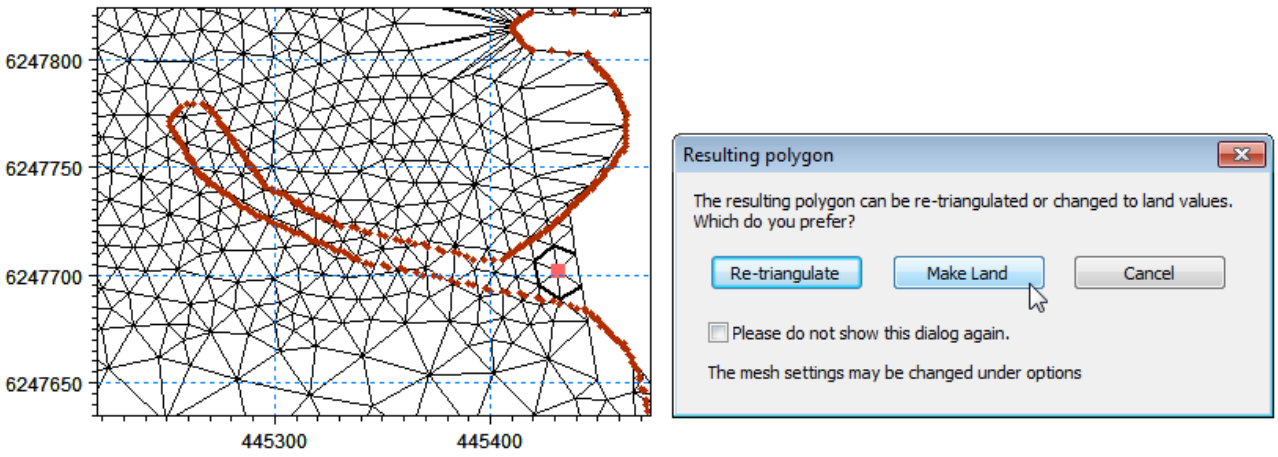


Figure 9.9 Deleting mesh nodes from existing mesh bathymetry

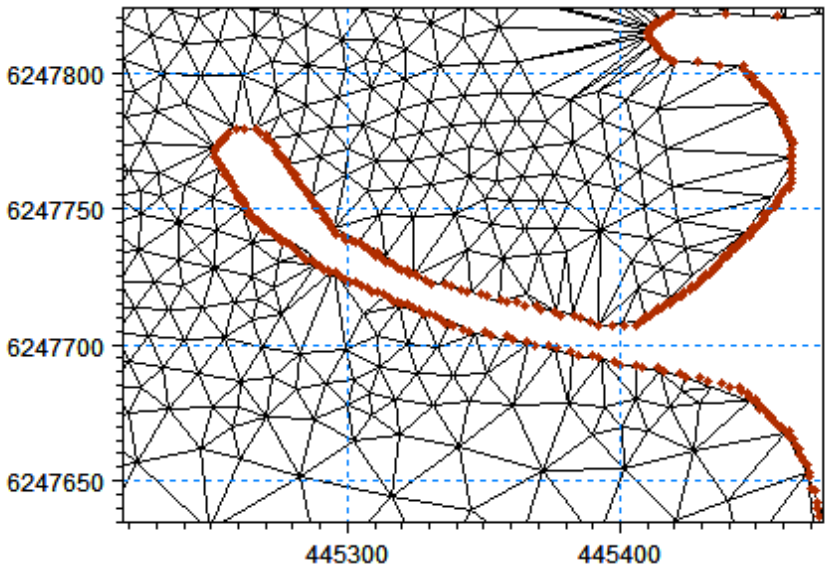


Figure 9.10 Resulting mesh after deleting and moving mesh nodes

Once the new boundary has been updated to a satisfactory level, the mesh can be optimised by using re-triangulation for a selected area.

9.1.5 Interpolating bathymetry

The node points already contain bathymetry information from the original mesh. However, as the points have moved around, and additional points have been added, it is necessary to interpolate the bathymetry values in the new mesh.

Add the original mesh as scatter data points using either the generated xyz-file with the original node points or the .mesh file itself. The display of scatter data shows where the new mesh match the old, cf. Figure 9.11.

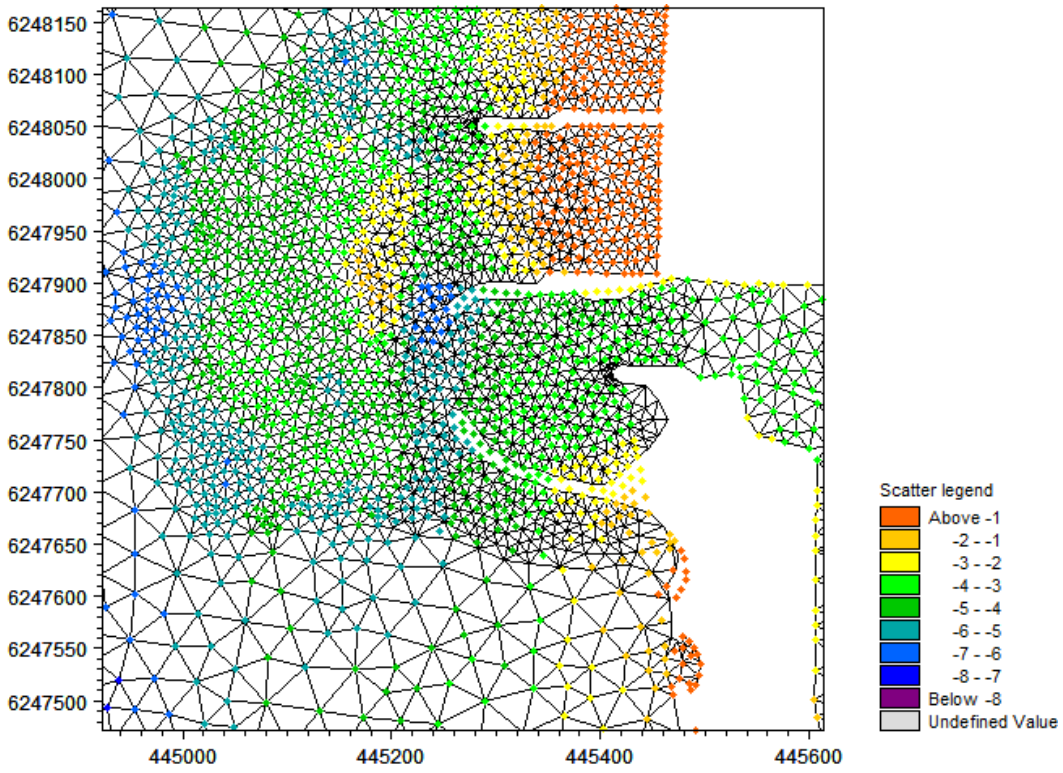


Figure 9.11 Scatter data from existing mesh displayed on new mesh

The resulting mesh have the same values in the node points that are not moved, whereas new and modified points will have bathymetry values interpolated from the original mesh as shown in Figure 9.12.

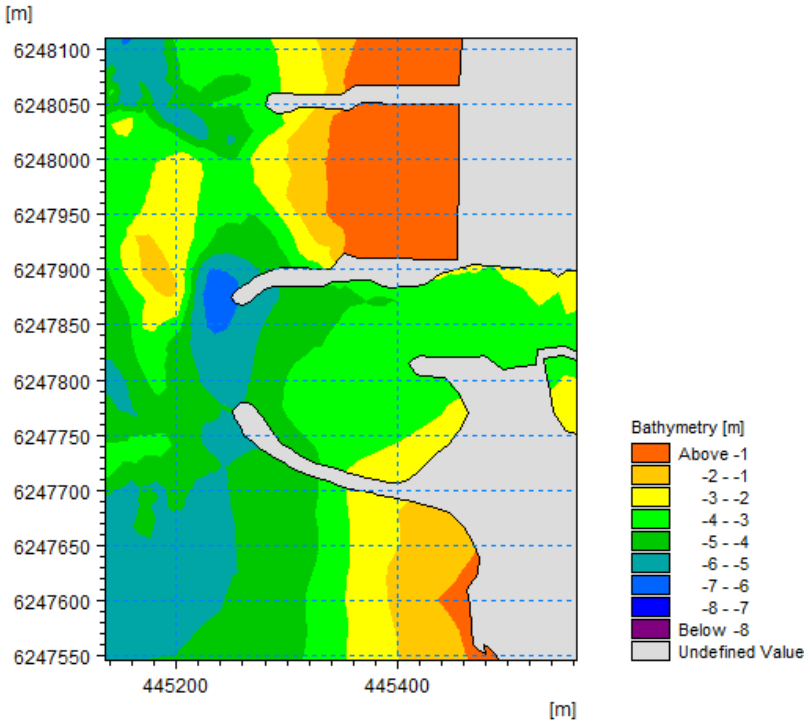


Figure 9.12 Detail of resulting bathymetry after modification and interpolation

9.1.6 Modifying properties of single mesh node

Note that the bathymetry in the southern part of the harbour entrance area has been interpolated based on the sediment accumulated south of the old structure. It is possible to modify the bathymetry values in several ways. A simple way is to modify the bathymetry in individual mesh nodes in the resulting mesh using the Data Viewer, but it is also possible to modify the mesh node properties for an individual node in the Mesh Generator by using the Edit mesh node option tool in the Mesh Editing toolbar as shown in Figure 9.13.

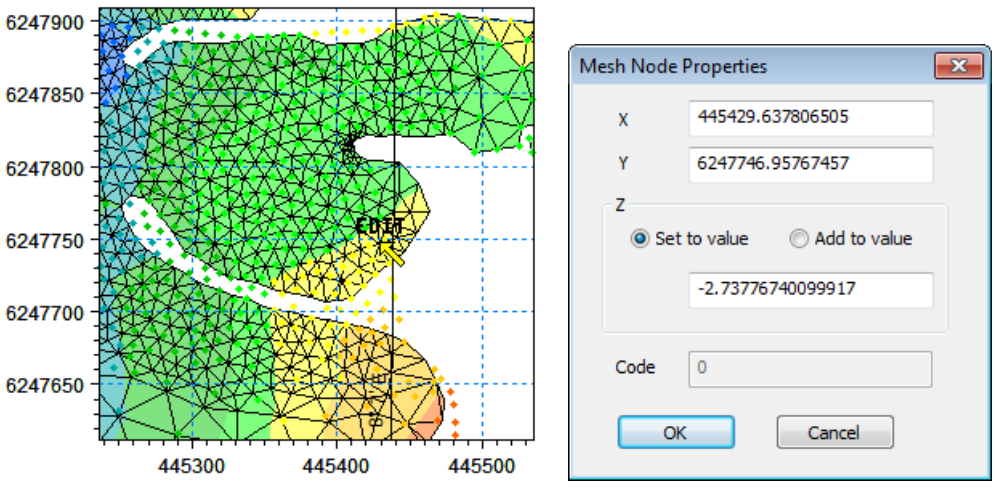


Figure 9.13 Mesh node editing using the Mesh Editing toolbar

9.1.7 Evaluating change

It is possible to see the changes in the bathymetry by subtracting the two files using the calculator in Data Manager. The resulting difference may be as shown in Figure 9.14.

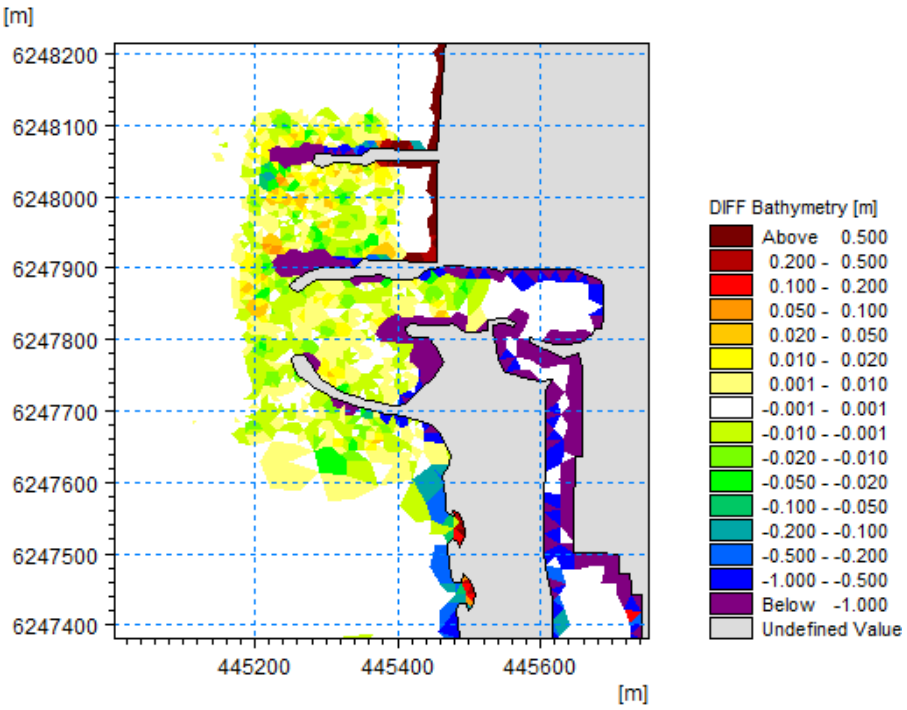


Figure 9.14 Difference in bathymetry values before and after modification. Note, the difference values near the land boundary lines are due to interpolation issues in the difference calculations

9.2 Use Re-established .mdf File for Existing Mesh

In case the number of changes to the mesh bathymetry is anticipated to be a major task, it may be less time consuming in the long run to re-establish (most of) the .mdf file used for creation of the existing mesh and then apply the changes using the Arc Editing facilities. Note that re-generating a mesh may lead to small changes in the location of the mesh nodes.

Create a new mesh generator setup by loading the existing mesh directly using the Import mesh command. Say **Yes** to generate information for Arc Editing. Change the settings from 'Mesh Editing' to 'Arc Editing' and save the mesh generator setup.

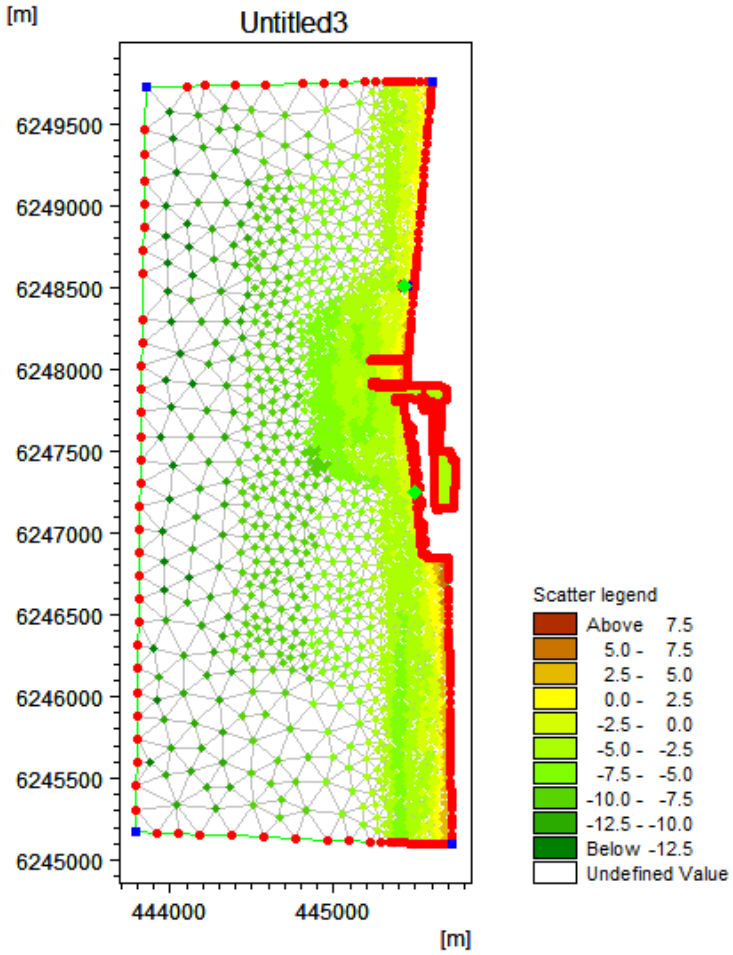


Figure 9.15 Outline of existing mesh bathymetry with auto-generated boundaries and scatter data

When a mesh is imported into Mesh Generator using the 'Import Mesh' functionality and you press **Yes** to generate information for Arc Editing, the program automatically sets the workspace and map projection and derives the position of the boundary arcs from the node point information. Additionally, the node points will be defined as scatter data in the setup. As the workspace and boundary outline have been defined it is now possible to use the generated .mdf file to make further modifications to the mesh as described in the previous chapters.

9.2.1 Import new boundary

As the layout of the harbour entrance has changed since the original mesh was created the next step is to import boundary data representing the new land contour. Import the boundary data derived from C-MAP (land_CMAP_F.xyz). Omit the display of the scatter data to allow for a clearer view of the imported boundary data.

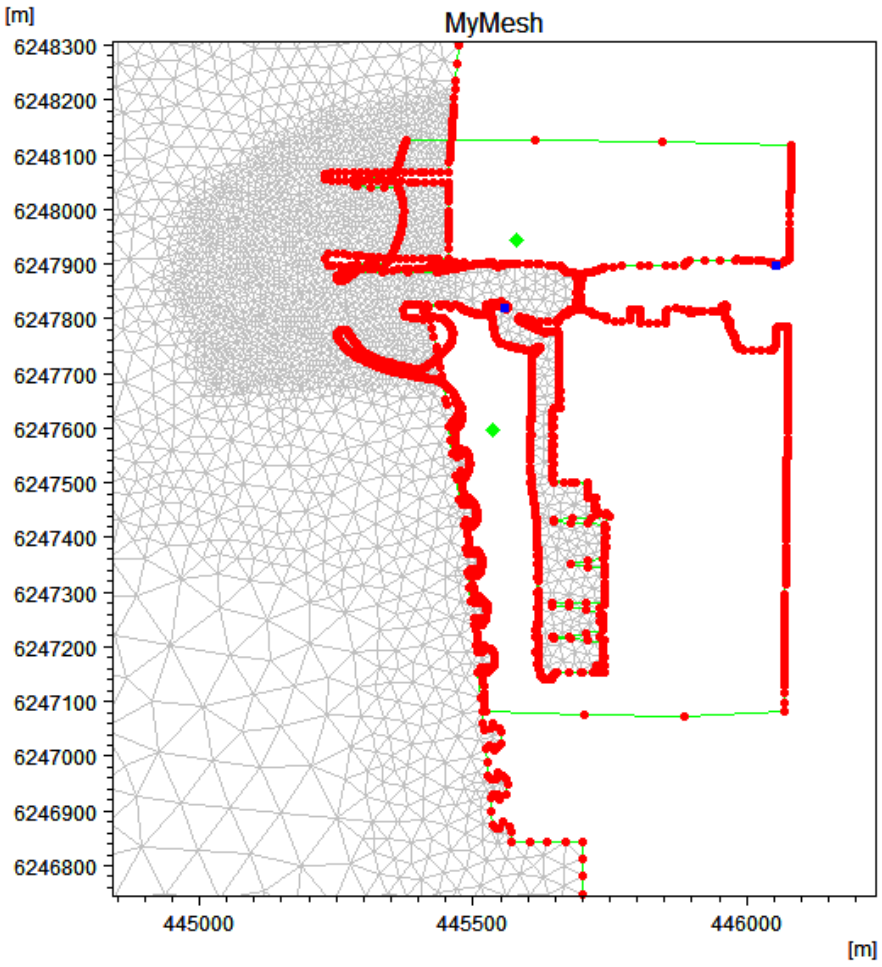


Figure 9.16 Imported land boundary together with original outline around harbour

Now modify the land boundary data around the harbour entrance to reflect the new layout of the piers. This can be done by introducing node points to split the boundary arcs, deleting sections of the old layout and connecting new relevant layout boundary arcs to the existing arcs away from the entrance.

After modifications and redistribution of vertices along the inserted arcs the resulting boundary may look as show in Figure 9.17.

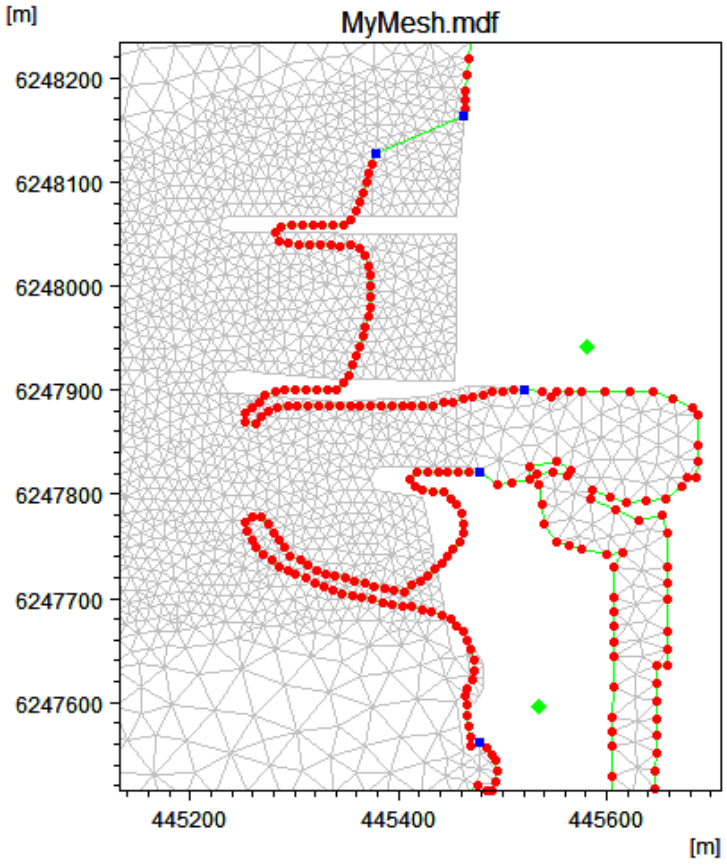


Figure 9.17 Resulting land boundary after modifications

9.2.2 Extract node point data from mesh

In order to re-generate the existing mesh as accurate as possible, it is necessary to define all the node points in the existing positions in the mesh generator setup.

The displayed scatter data from the imported mesh contains the xyz information of all the nodes (see Section 1.6).

These points can be converted to mesh nodes by using the Convert button in the Manage Scatter Data dialogue as shown in Figure 9.18

All the scatter data will now be converted into node points, apart from node points that is part of an arc. As node points are fixed locations, a mesh using the original mesh nodes will automatically be created during the mesh generation process, thus creating a similar mesh if it is purely triangular elements. In case of quadrangular elements in the original mesh, the new mesh will contain two triangles for each quadrangle.

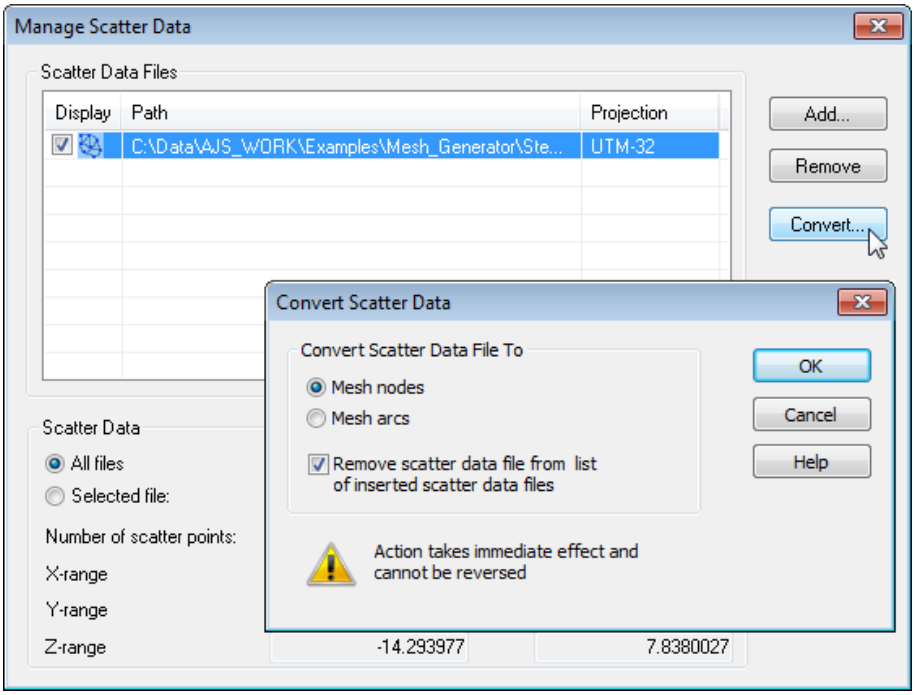


Figure 9.18 Converting scatter data to node points

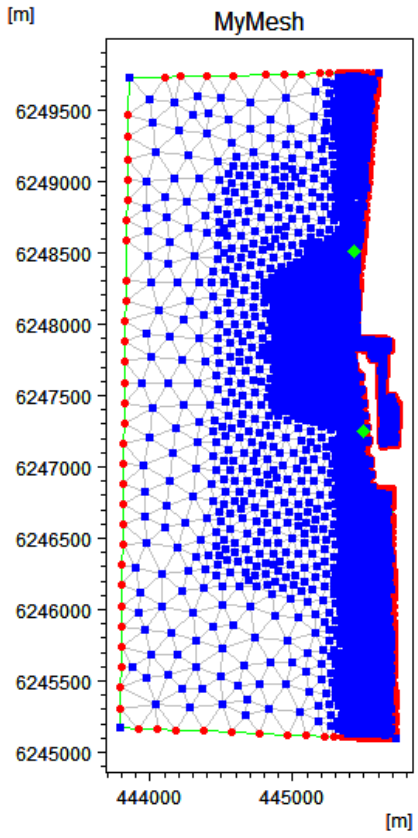


Figure 9.19 Existing node points derived from scatter data

Before creating the mesh the node points located within land or close to new boundaries should be deleted in order to avoid very small elements. The result of this process could look as shown in Figure 9.20.

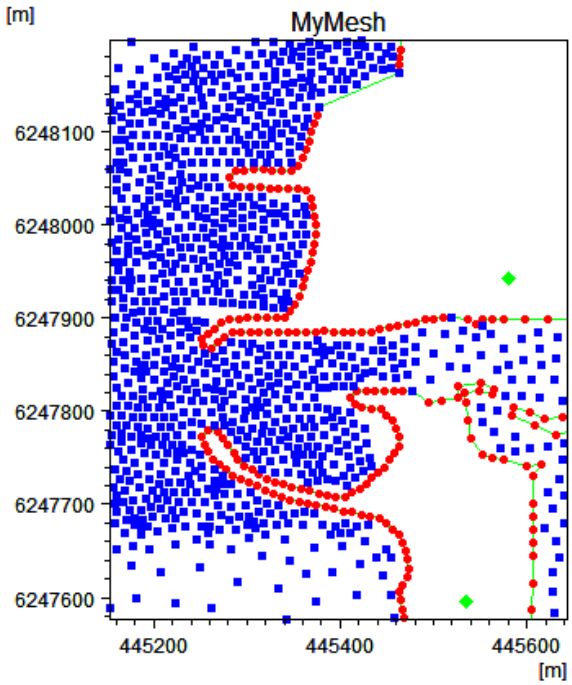


Figure 9.20 Setup after deleting node points (from scatter) near boundaries

9.2.3 Create mesh

After saving the setup generate the mesh. As the node points are fixed locations in the mesh generation process, a mesh very similar to the original existing mesh will automatically be created, even if the Maximum element area is larger than previously. If the Maximum element area is smaller than previously, the mesh may end up being more refined, but still including the original node points included.

Affter reviewing the existing mesh you may choose to include additional scatter data points and re-generate the mesh in order control the mesh generation even more.

Note that the node points in the mesh may move if the generated mesh is smoothed.

9.2.4 Interpolate bathymetry

Next, import the scatter data points or alternatively the original mesh as scatter data and interpolate the bathymetry values in the mesh. In this case the interpolation option ‘Set value from scatter data’ will make a quick interpolation finding the exact same bathymetry values as in the original mesh.

The resulting mesh will be as shown in Figure 9.21.

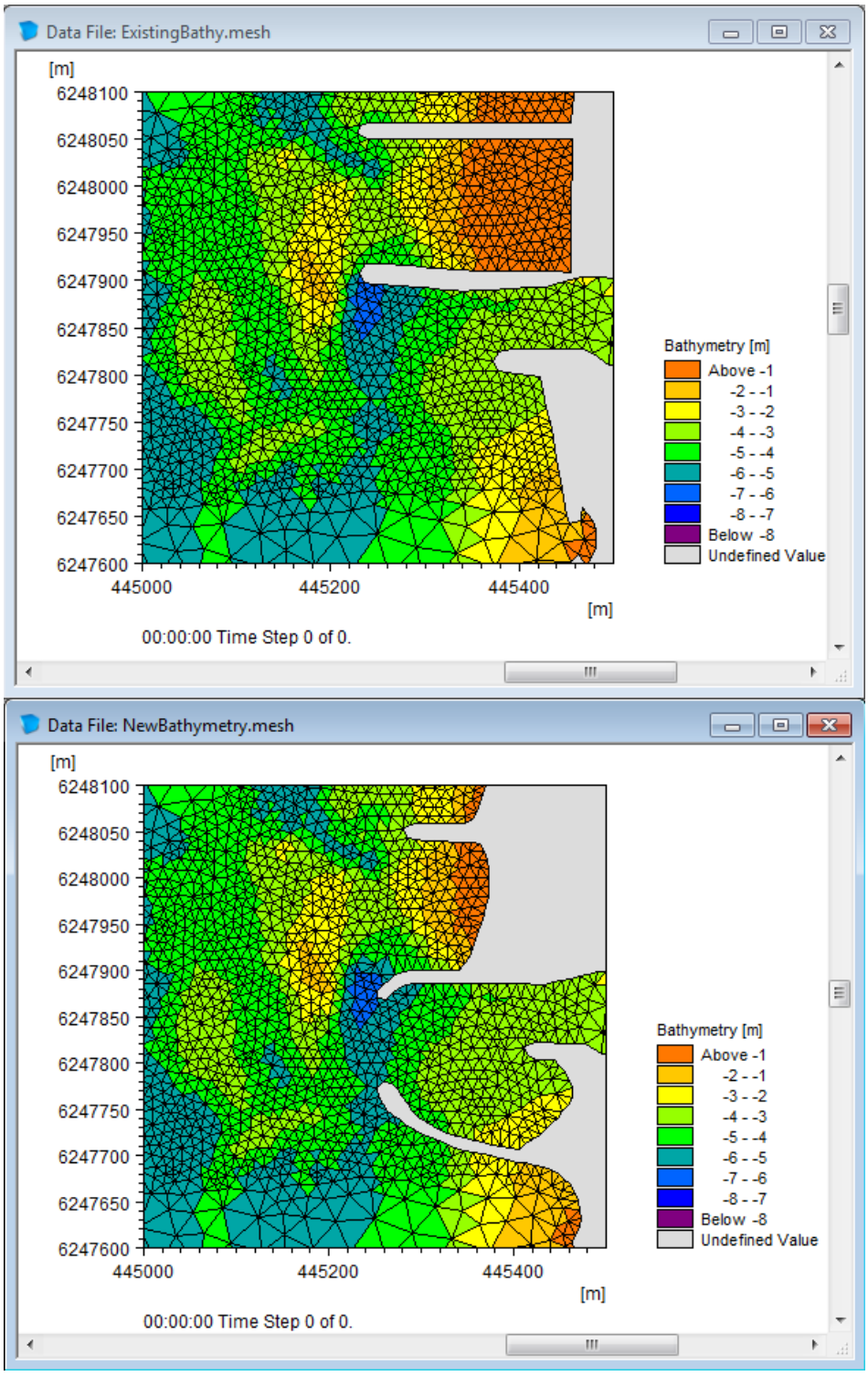


Figure 9.21 Resulting mesh bathymetry. Upper: original mesh, Lower: new mesh