Preprocessor Janet

First Steps

Martin-Edition

smile consult GmbH
www.smileconsult.de

Version 1.0 (10.12.2005)

Getting Started

1

1.1 Graphical User Interface

Starting Janet displays its main window. The main window consists of different parts, which are illustrated in Figure 1-1. The working window displays the models (computational grid, digital terrain model, density function, etc.) with various visualization methods. All mouse-based editing is performed in this part of the graphical user interface. The working window also allows zooming with no limiting zoom factor. The current zoom state is highlighted in the overview window.

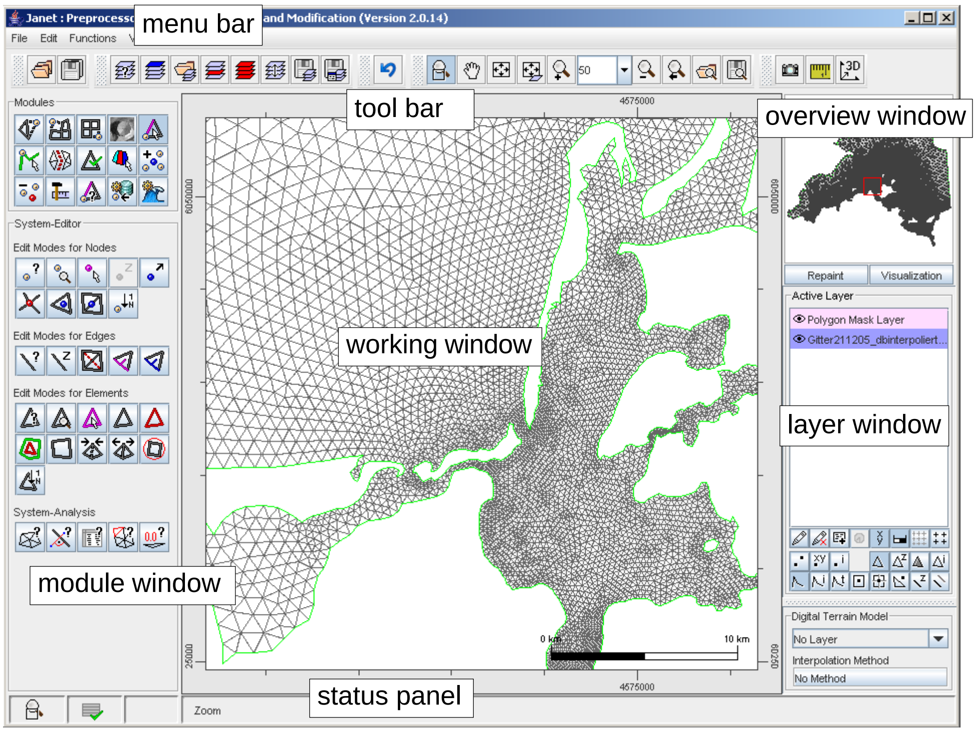

Figure 1-1. The graphical user interface

The tool bar gives access to the basic functionality. Among the offered functions, there are basic file operations, the undo function, zoom operations and the 3D visualization. More model specific functions are available in the module window. Various functions grouped in different modules allow creating, editing, optimizing and analyzing different kind of models. The currently choosen function is displayed in the status panel with its icon and

a short description.
In the layer window the currently edited model is selected. The layer window also allows quick editing of basic visualization settings for the working area. Finally, the menu bar offers access to file history lists, basic program preferences and the help system.

1.2 Loading a Mesh

A mesh is imported in the preprocessor with the [Load Layer] function from the tool bar or alternatively with the [Load Layer] → [Choose File] menu item in the file menu.
A file (e.g. “u_channel.jbf” from the directory “examples”) is choosen in a file selection dialog. The import interface supports various file formats which are presented by different file filters. For the given example the file filter “Janet Binary Format” and the file “u_channel.jbf” are selected.

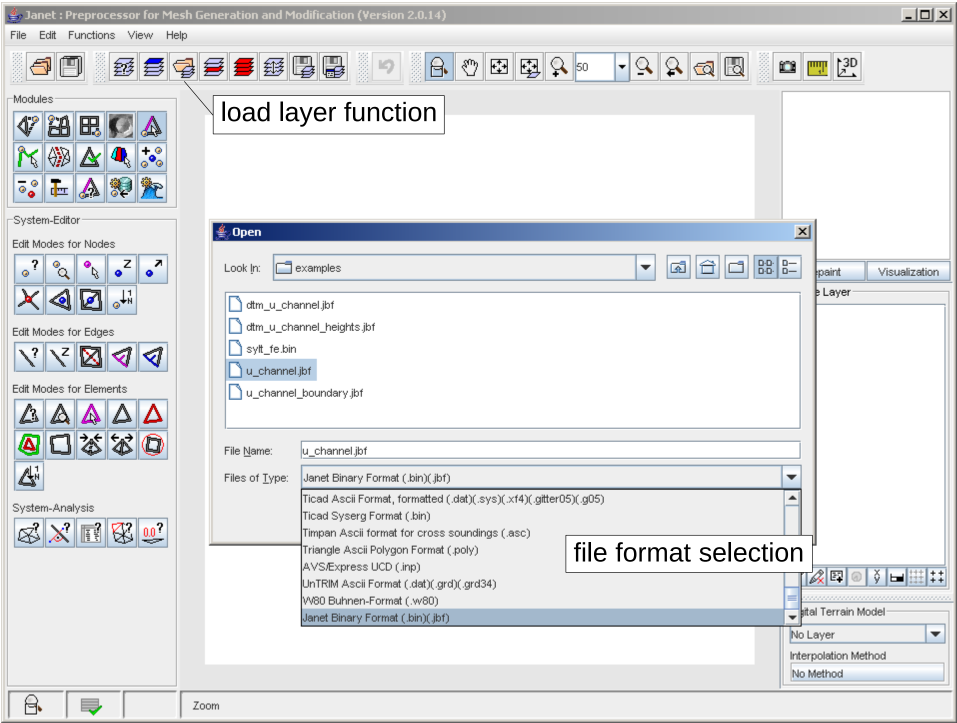

Figure 1-2. Loading the file “u_channel_be.sel”

Having successfully selected the file, the user is asked to define the model/layer type. In this specific case the type “Finite Element Mesh” is preselected and can be accepted without changes.

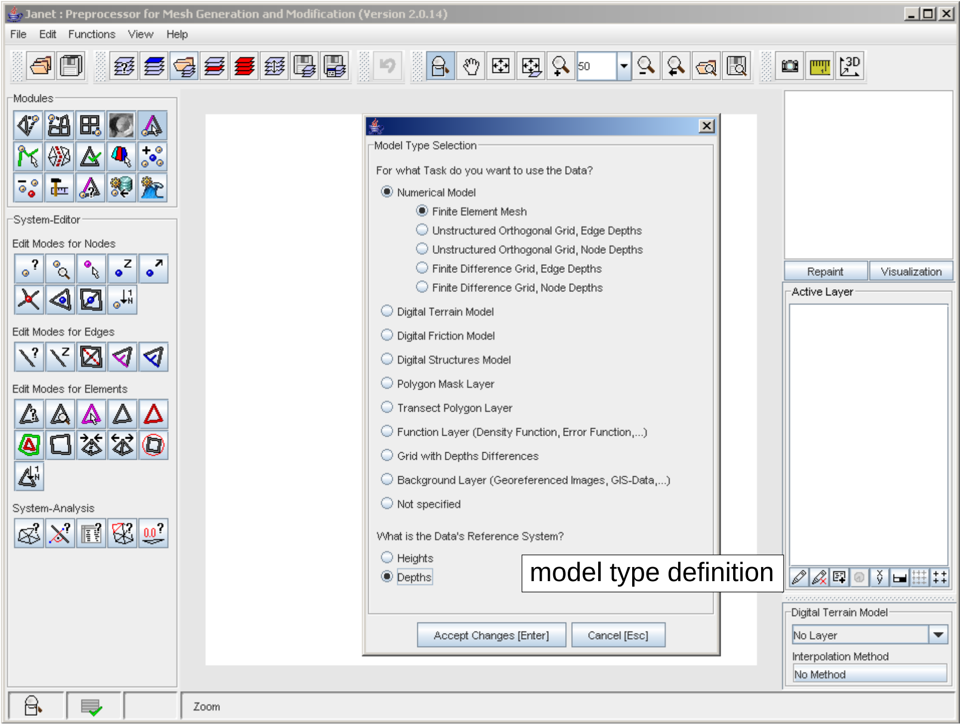

Figure 1-3. Model type definition

1.3 Using the Zoom Function

After the grid was imported into the preprocessor, the entire domain is displayed in the working window. The next task is to change the zoom factor to a more detailed view of the grid.

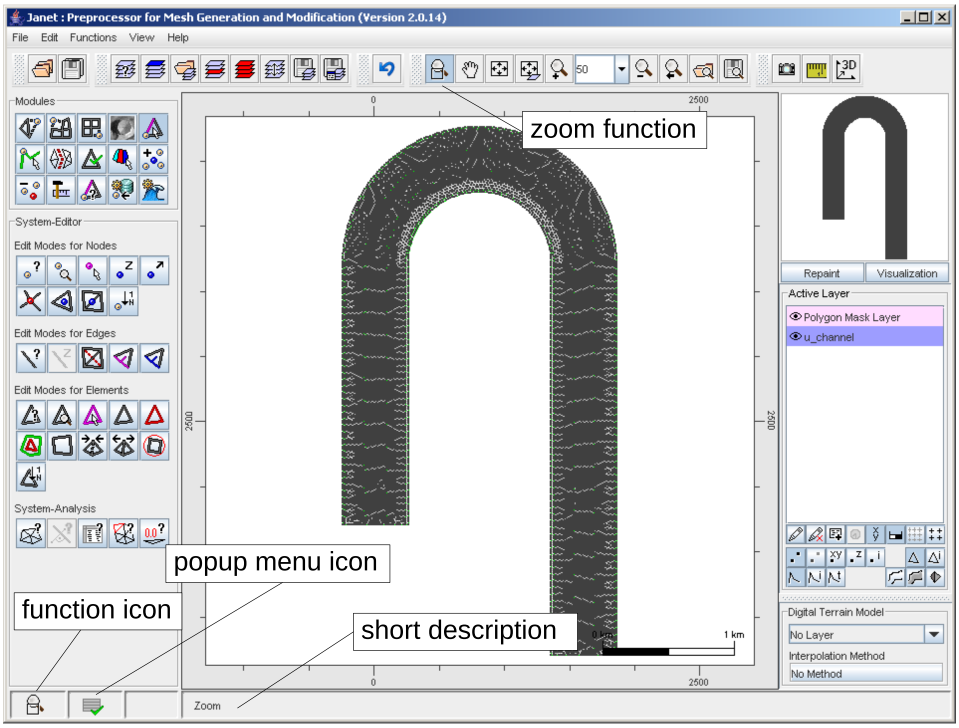

Figure 1-4. Main window after the file import

The zoom function is enabled by selecting the specific button in the tool bar. Zooming is a mouse–based function, so the button remains pressed and the status panel is updated with the current settings. Besides the function icon and a short description, the popup menu icon indicates that a popup menu with further available actions can be accessed via the right mouse button on the working window.
Zooming is done with the left mouse button by pressing and dragging a bounding box for the desired zoom state.

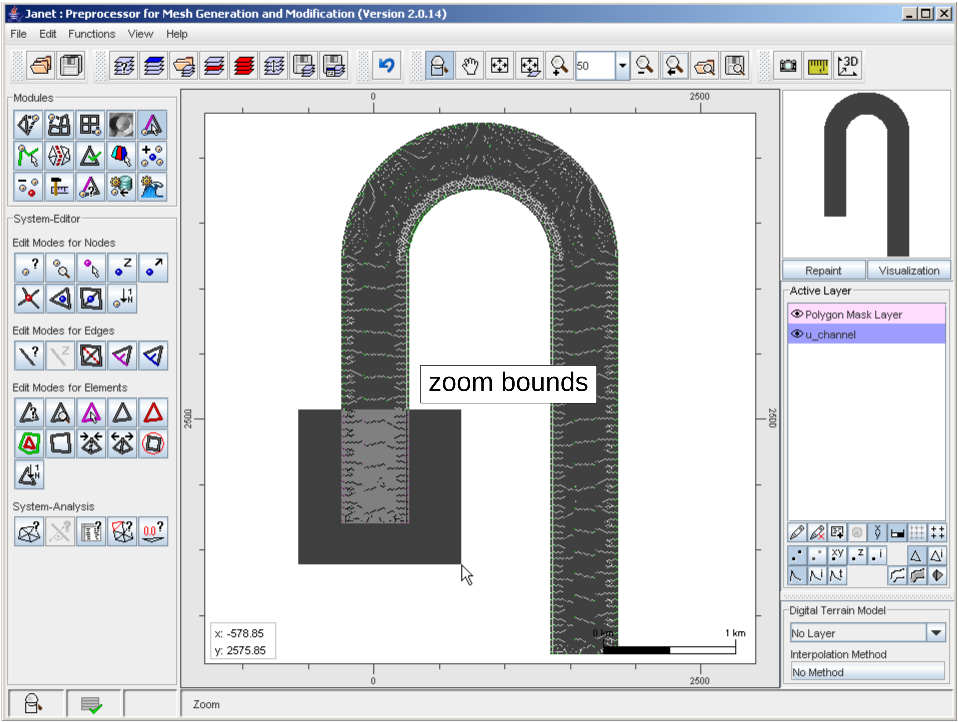

Figure 1-5. Changing the zoom state

When the left mouse button is released the working window is repainted for the modified zoom state.

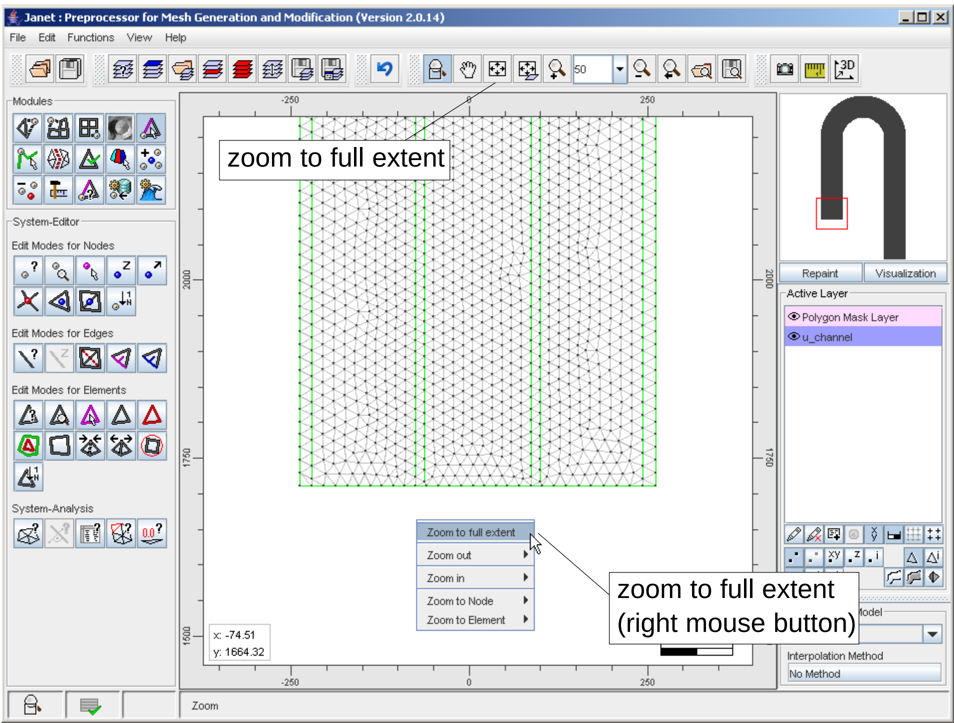

Figure 1-6. Zoom to full extent

To restore the default zoom state, either the menu item “Zoom to full extend” in the popup menu of the zoom function (right mouse button) or the function from the tool bar has to be selected (see Figure 1-6).

1.4 Modify a Grid’s Visualization Settings

The preprocessor allows various visualization methods for finite element meshes. Basic modifications can be performed with the quick edit buttons in the layer window.

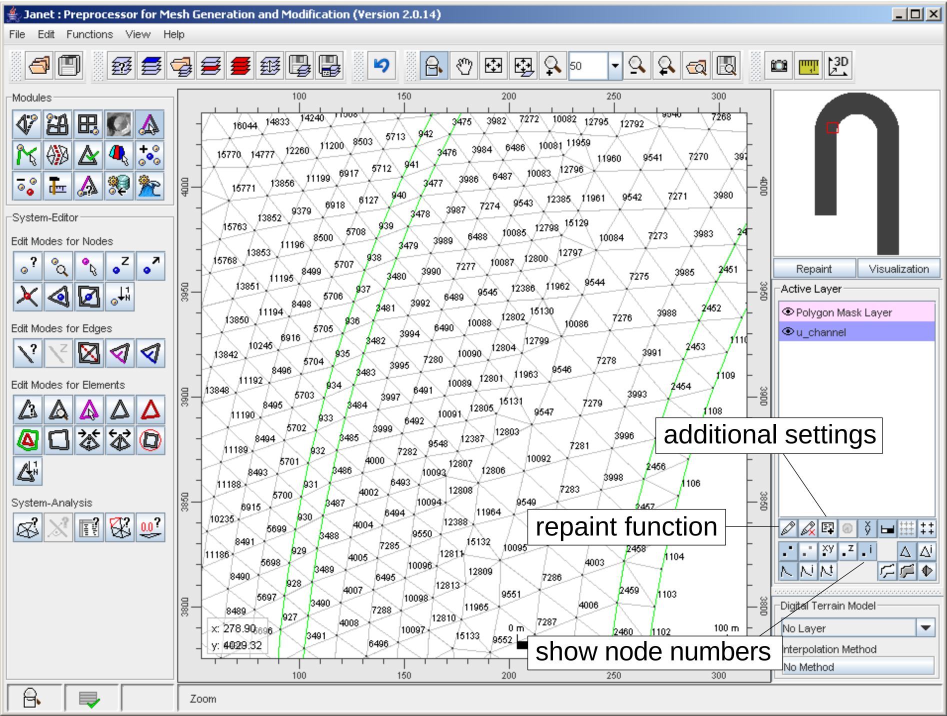

Figure 1-7. Modify visualization settings with quick edit buttons

To display the node numbers for example, the specific button (see tooltips!) is selected and the changes are submitted by pressing the repaint button.

Additional settings can be modified in a separate dialog window. The user interface offers different methods for alphanumerical plotting the grid’s properties, enabling 2D surface plots and displaying the basic geometric objects of the model.
A contour plot for example is performed by activating the tab “Surface” and selecting the Checkboxes [Show Flooded Contours].

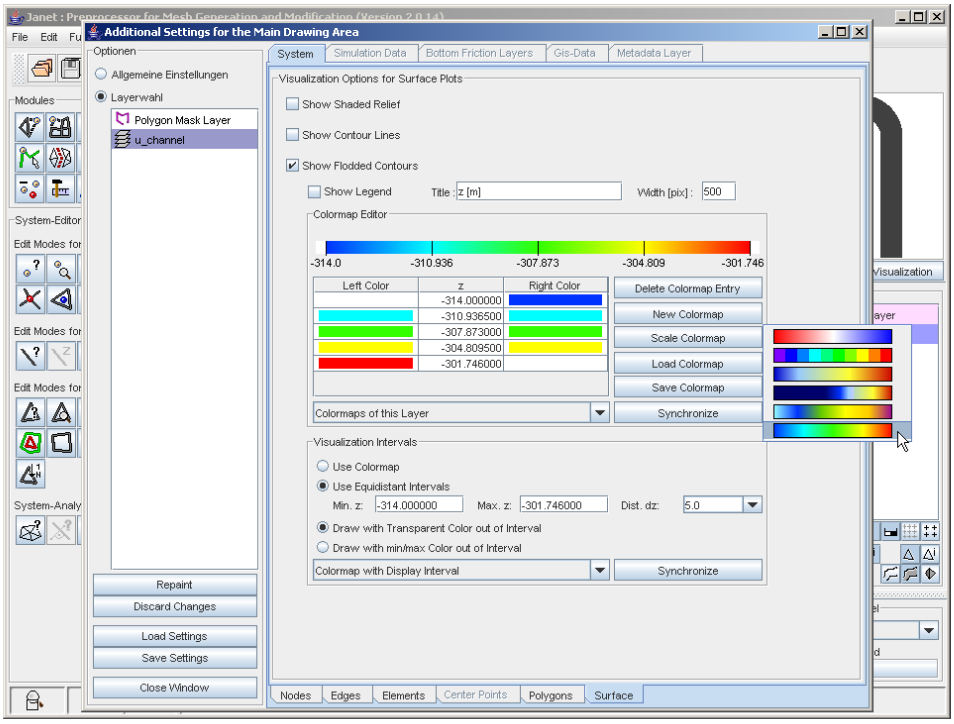

Figure 1-8. Show a surface representation of the grid

The surface plot relies on a colormap which assigns colors to depths values. The colormap can be individually modified with the integrated editor. The editor allows operations to create map entries, to perform an automatic scaling to the z-bounds of the grid and offers a set of predifined colormaps. The application of this visualization method for the "u_channel" mesh leads to the result presented in the next figure.

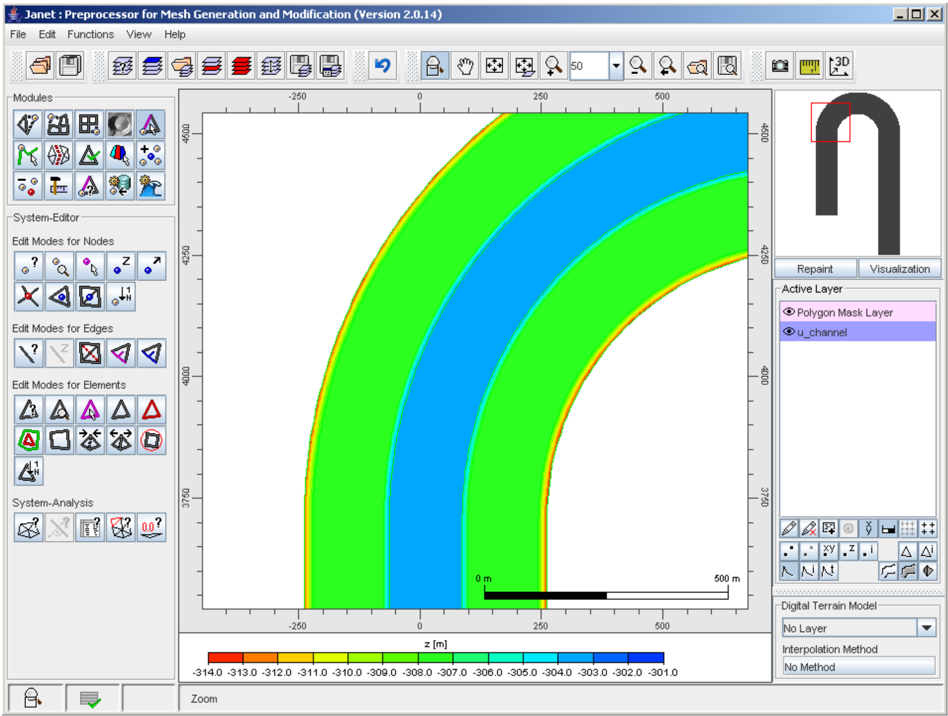

Figure 1-9. Surface representation of the grid

1.5 3D-Visualization

The integrated 3D viewer allows a detailed analysis of the finite element mesh. The 3D function of the tool bar can be applied to the current zoom state, so even large models are comfortably visualized. Different operations, such as rotation, scaling, etc. can be performed with the 3D viewer.

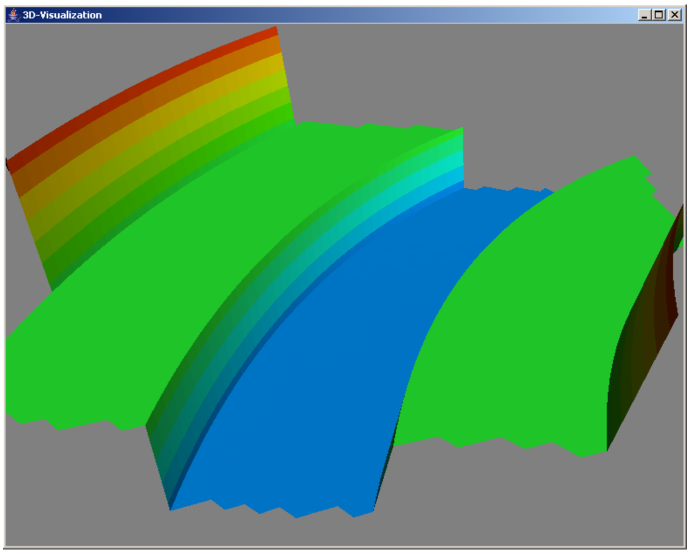

Figure 1-10. 3D viewer

Creating a simple Mesh

2

2.1 Basic Steps

In this chapter basic steps to create a first finite element mesh for Marina are presented. The files “u_channel_boundary.jbf” and “dtm_u_channel.jbf” from the “expample” directory are used to demonstrate

- meshing a boundary polygon,
- using a refinement method to generate a discretization,
- mapping bathymetry data to the mesh,
- defining open/land boundaries and finally
- exporting the mesh.

The simple example touches some basic concepts of the preprocessor. It helps to understand

- the integration of digital models (e.g. terrain model, bottom friction model) in the grid generation process,
- the layer handling and
- using masks for algorithmic editing.

The basic concepts shown in this example can easily be transferred to more complex problems.

2.2 Meshing a Boundary Polygon

The “examples” directory contains a file “u_channel_boundary.jbf” in the “Janet Binary Format”, which is imported into the preprocessor with the [Load Layer] function (see chapter 1).
Figure 2-1 shows a detailed view of the imported polygons. Besides the closed boundary polygons, the file contains a set of additional polylines which serve as geometric constraints in the mesh generation process. The polygon points define the mesh nodes for the initial triangulation whereas the polygon segments are used by the triangulation algorithm to generate “constraint edges” (predefined edge configurations) in the resulting grid.

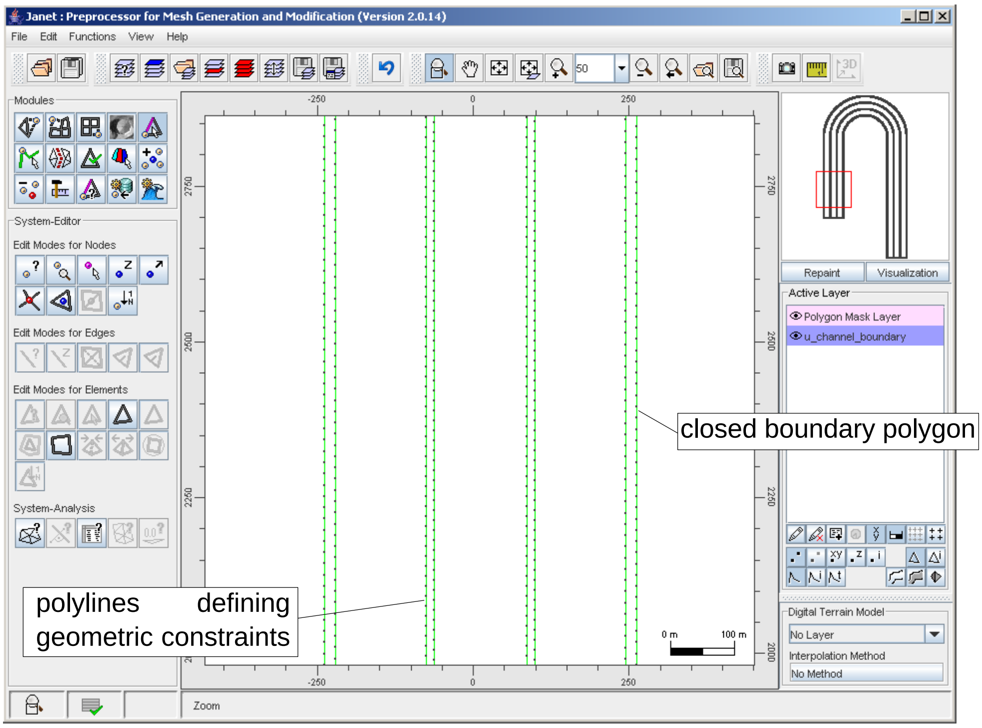

Figure 2-1. Boundary polygon and additional polylines

All triangulation functions are grouped in the “Triangulation Module” which can be found in the module section of the main window. The function “Triangulation with Boundary Polygons and Constraint Polygon Segments” is then started to generate the initial discretization.

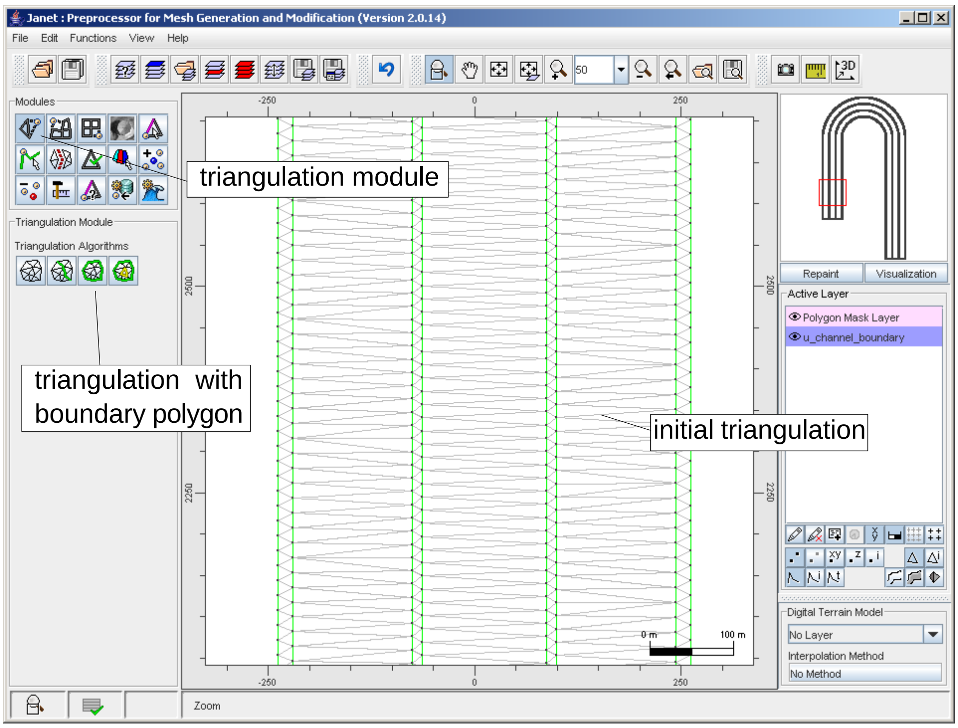

Figure 2-2. Initial triangulation

2.3 Application of a Refinement Method

The preprocessor contains various refinement methods to gerate adequate discretizations for finite element methods. The refinement module offers access to

- Criterion Based Refinement (barycentric mesh refinement with a refinement condition)
- Advancing Front Refinement
- Grid Refinement (refinement of an unstructured mesh with regularely generated nodes)

In this example, the "Advancing Front Refinement" is applicated to the initial triangulation (Figure 2-3).

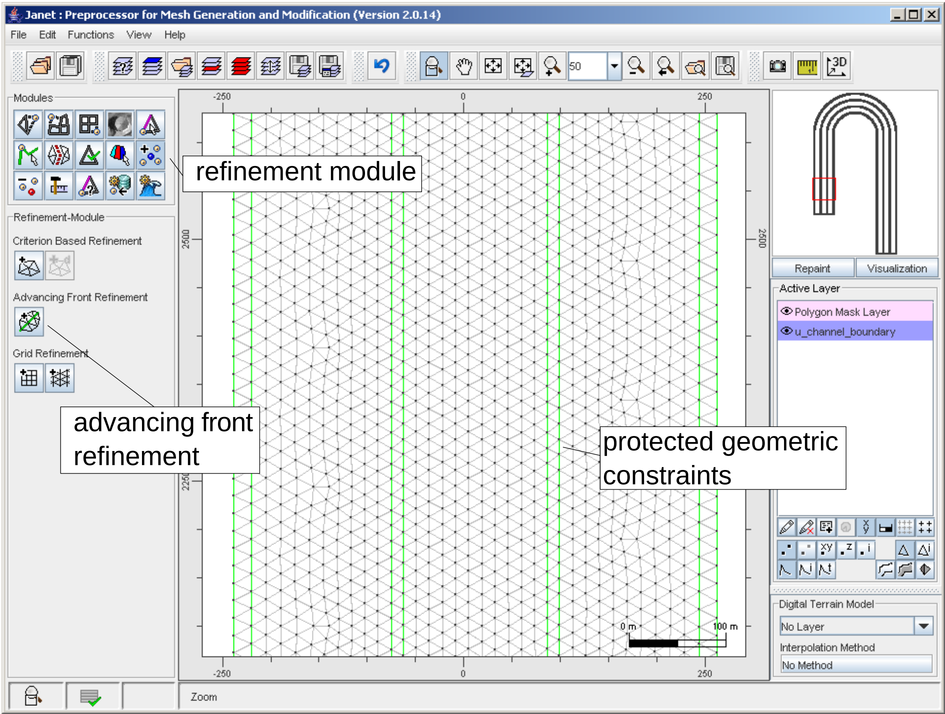

Figure 2-3. Refined mesh

2.4 Mapping Bathymetry Data to the Mesh

Bathymetry data combined with an approximation method represents a digital terrain model in the concept of the preprocessor Janet. Various approximation and interpolation methods are available (e.g. linear interpolation, nearest neighbour interpolation, natural neighbour interpolation, inverse distance interpolation) and can be applied to unstructured and gridded bathymetry data. In the following example a linear interpolation on triangulated data points is used for mapping data to the mesh.
The “examples” directory contains an unstructured grid “dtm_u_channel.jbf” in the “Janet Binary Format”, which is imported into the preprocessor with the [Load Layer] function (see chapter 1). The layer type is changed to “Unstructured Digital Terrain Model”, so this layer is automatically set as the digital terrain model in the layer window of the graphical user interface. The layer is also added to the layer list.

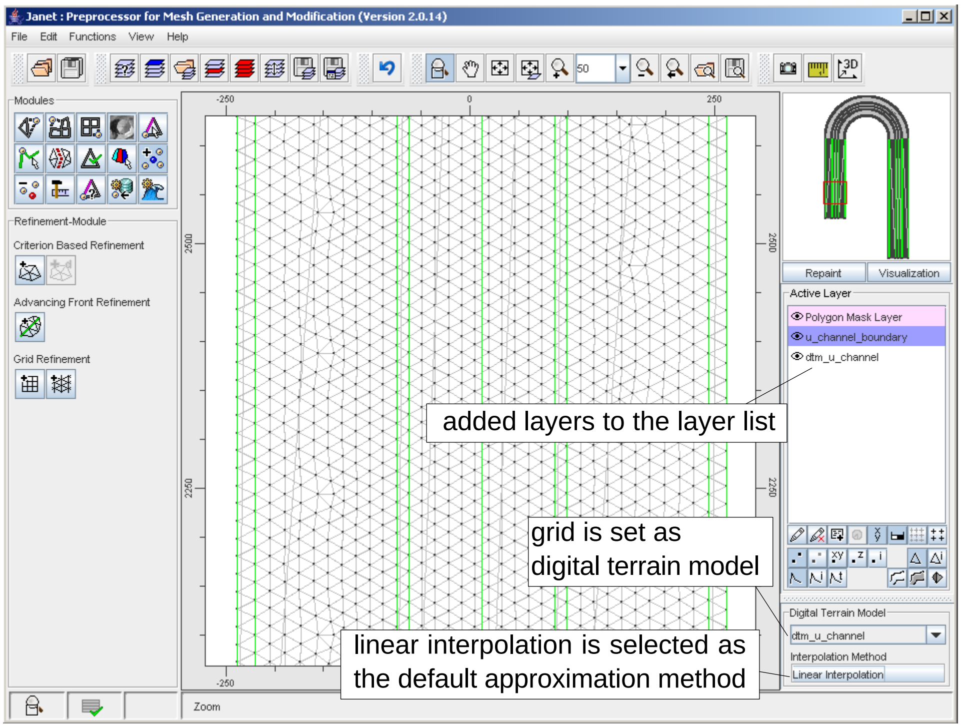

Figure 2-4. The imported digital terrain model

The preselected approximation method is edited by pressing the button labeled [Linear Interpolation]. All available methods and their related parameters are presented in a dialog window. For this example, the additional option [Allow Extrapolation on a triangulated DTM] is selected for the linear interpolation method.

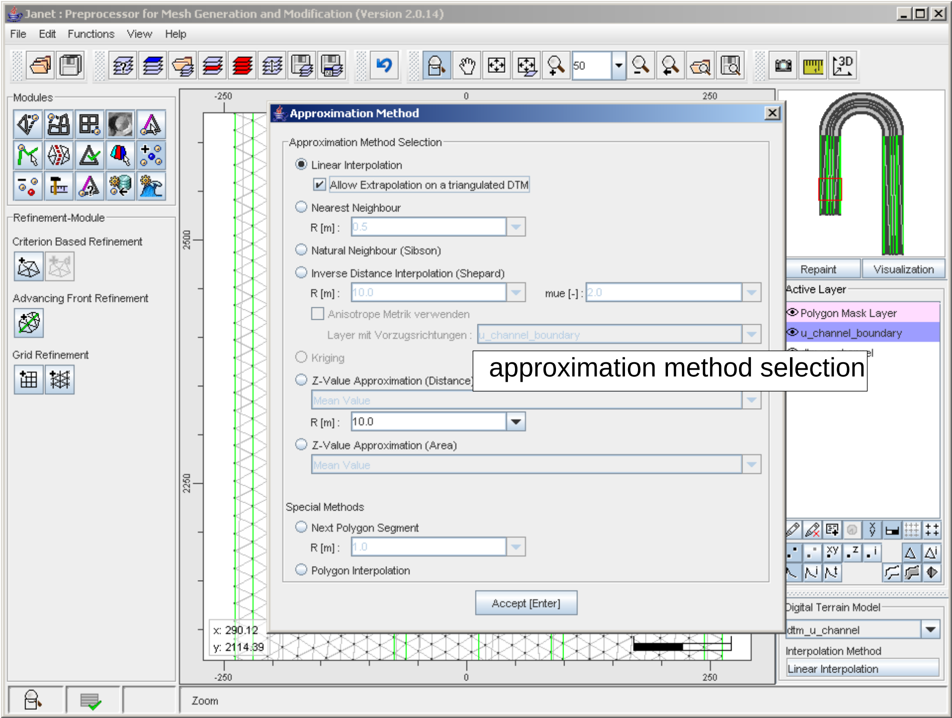

Figure 2-5. Editing the parameters of the approximation method

In a next step the depth values of the layer's "u_channel_boundary" nodes shall be interpolated on the digital terrain model. All modifications performed with the preprocessor's functionality are applied to the layer indicated as the "active layer" (highlighted entry in the layer list). To change this layer just select the specific layer with the mouse in the layer window. For the interpolation example the active layer remains the layer named "u_channel_boundary".

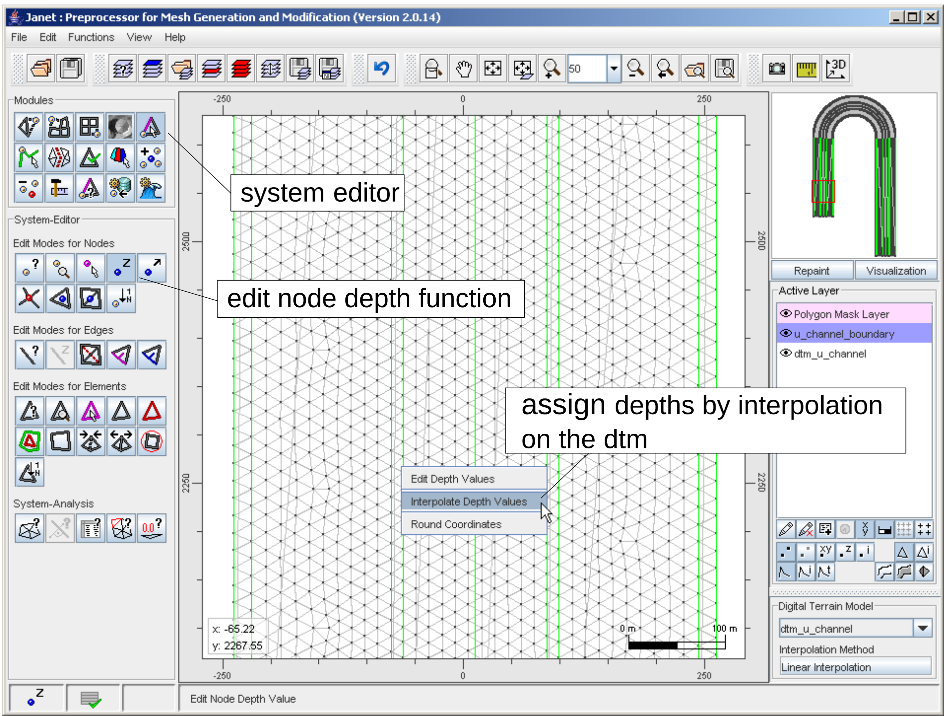

Figure 2-6. Starting the interpolation

The function “Edit Node Depth Value” in the system editor enables the editing of nodal z-values. The mouse function can be used to edit single values by interactively selecting a node on the working window. Furthermore, in a popup menu, different algorithmic editing functions are offered. The popup menu is accessed with the right mouse button pressed in the working area. Choosing the menu item [Interpolate Depth Values] opens a dialog window where the parameters for the interpolation are edited.

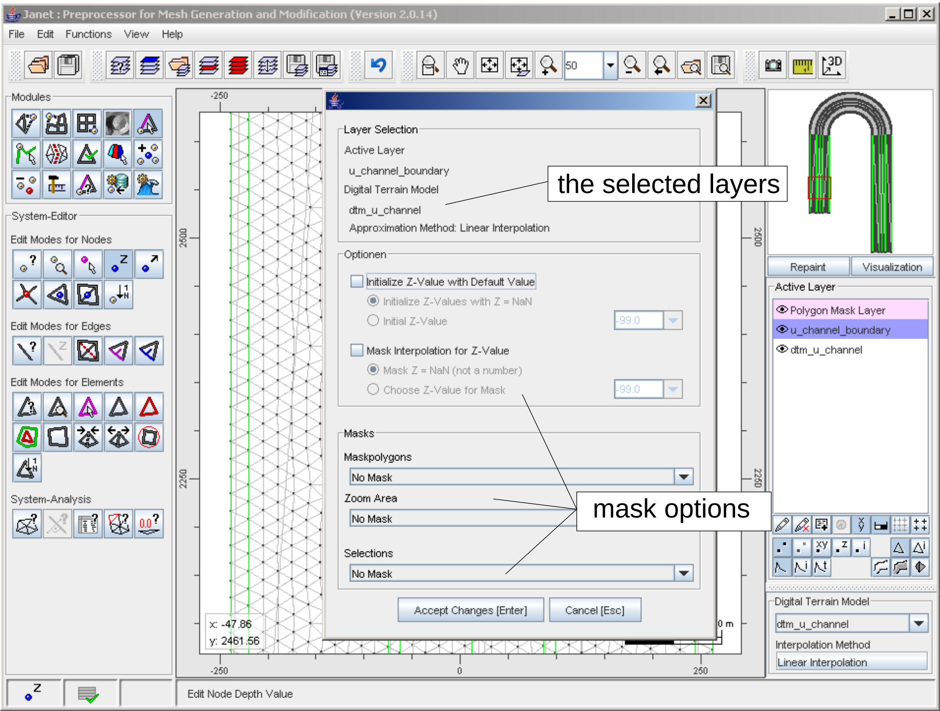

Figure 2-7. Dialog window to modify the options for the interpolation

In the example no modifications are applied in the dialog, so the depths of all nodes in the entire grid are interpolated. The button [Accept Changes] starts the interpolation, the progress is plotted in the status panel and finally all views are updated when the method finishes.

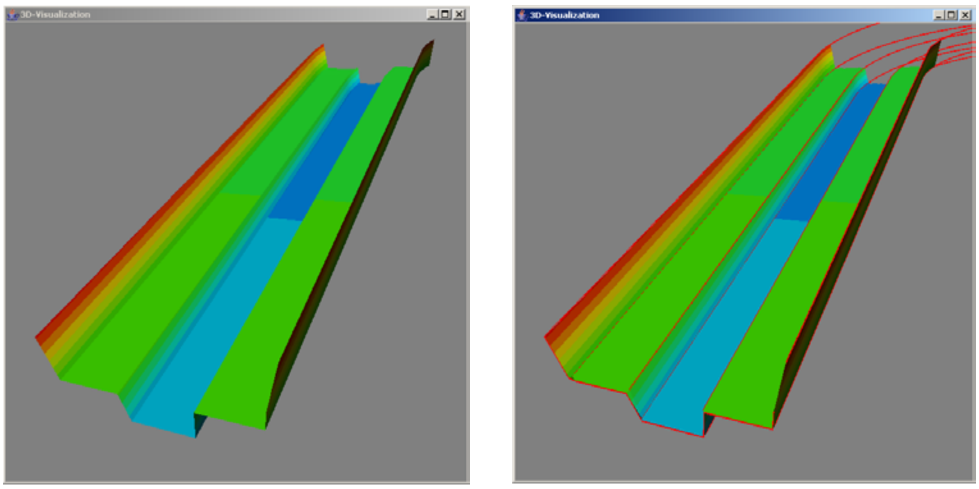

Figure 2-8. 3D view of DTM (left) and grid with interpolated depths (right)

The result of the interpolation is illustrated in Figure 2-8. All integrated polylines in the computational grid are plotted in the 3D visualization.

2.5 Definition of Open/Land Boundaries

The definition of open/land boundaries is done by setting markers to those edges which define the land boundary. The function [Edit Boundary Markers] of the system editor provides this functionality.

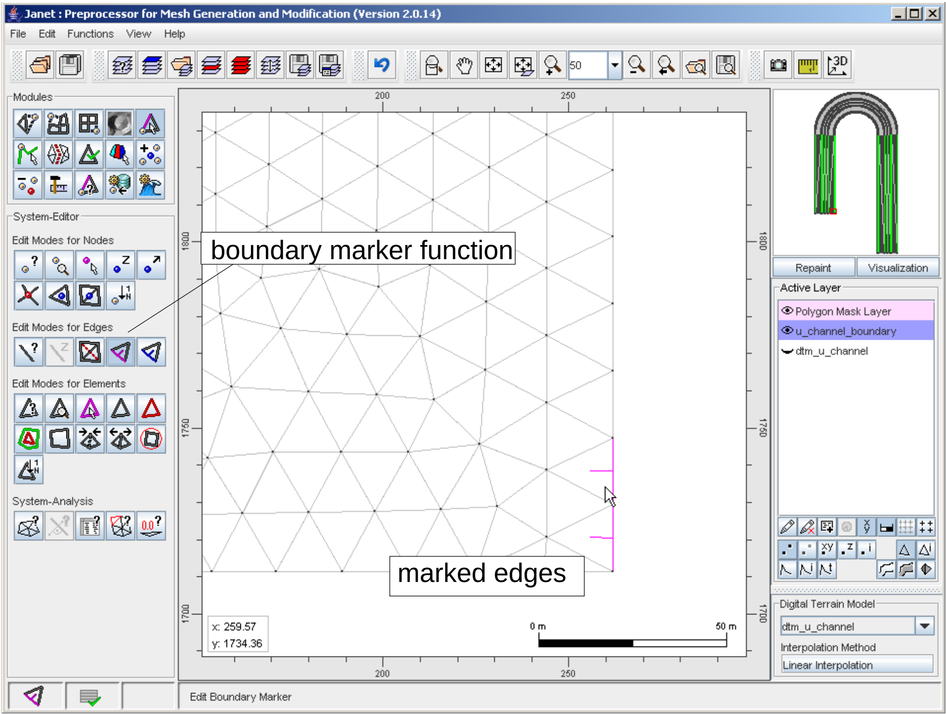

Figure 2-9. Marking edges with the mouse

Single edges are marked by selecting the geometry object in the working window. The current zoom state can be panned by pressing the [Alt] key and dragging the left mouse button in the desired panning direction. The boundary marker functions still remains active.
The presented approach occurs to be time consuming, even for a simple model. In a next step a more automatic way is suggested. The popup menu on the right mouse button offers the function [Set Boundary Markers] which is started and confirmed in the dialog window. All boundary edges are marked with this function as it can be seen in the next figure.

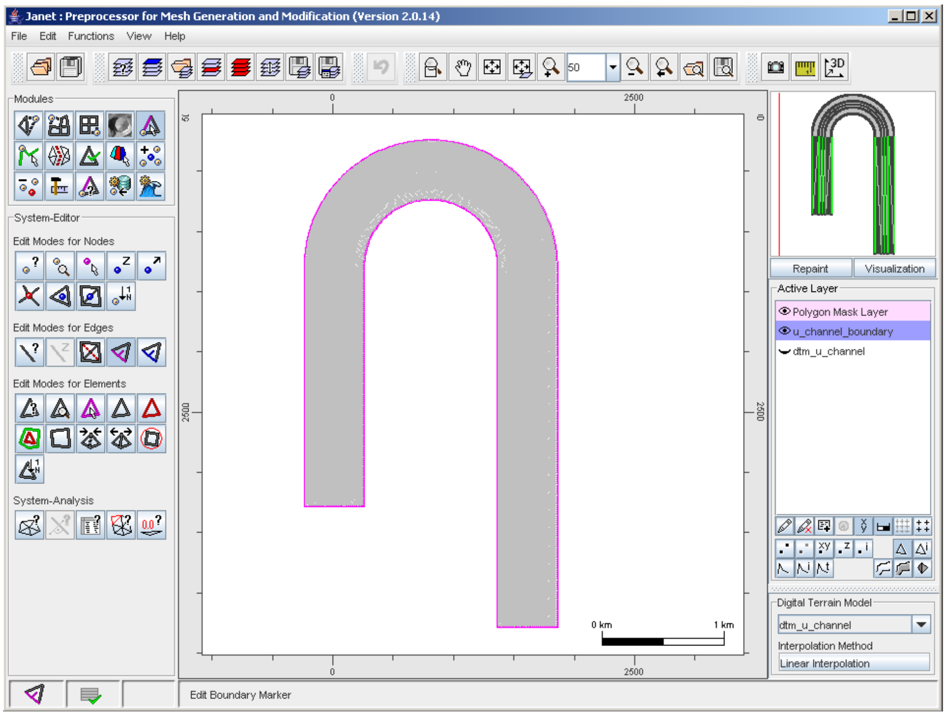

Figure 2-10. Automatically set boundary markers

The applied markers at the up- and downstream boundary have to be deleted again. For this task the preprocessor's masking functionality is used. Closed polygons can be defined on a special layer. The areas described by the polygons are then used to limit the application of an algorithmic function. To active this layer, the entry "Polygon Mask Layer" in the layer list has to be selected.
Polygons are created with the [Define Polygon] function in the polygon editor. Figure 2-11 shows one of the two generated polygons for the example.

After the polygon creation is finished, the active layer is reset to "u_channel" and the function [Edit Boundary Markers] is again selected. Markers are deleted with the [Delete Boundary Markers] method from the related popup menu. To enable the polygon mask, the option "Edge completly inside a maskpolygon" is choosen in the Maskpolygon section (Figure 2-12).

Starting the editing function finally leads to the desired result (Figure 2-13).

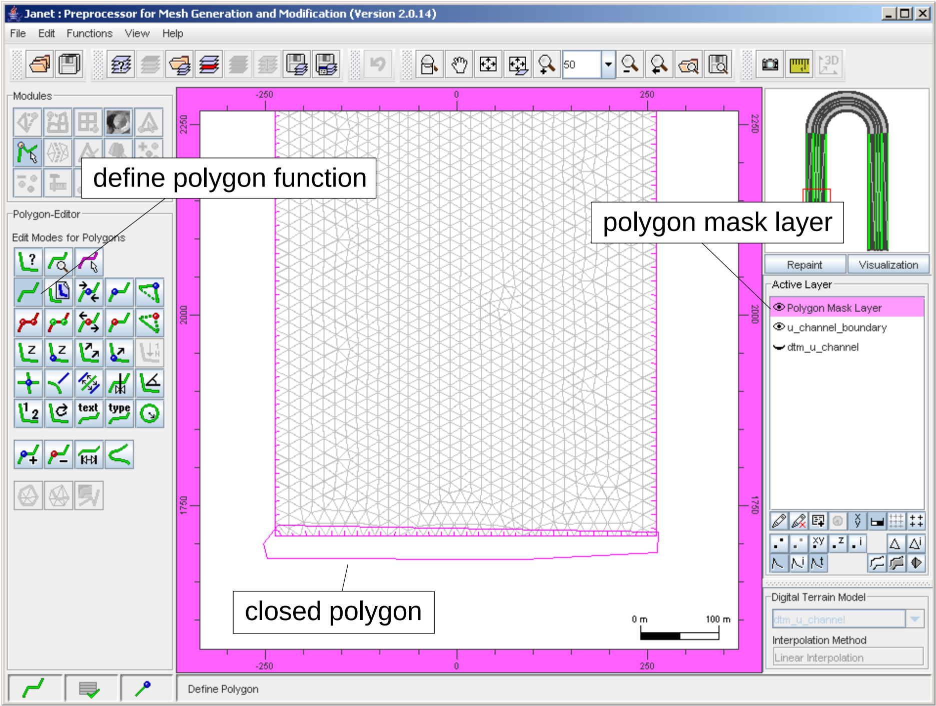

Figure 2-11. Defining a polygon mask

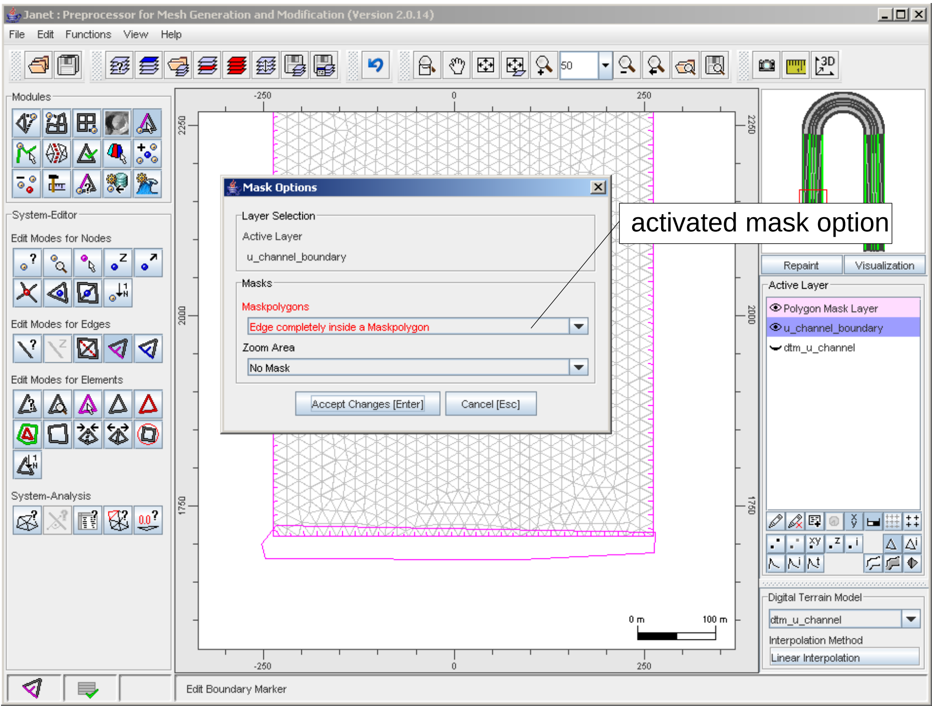

Figure 2-12. Editing the mask settings

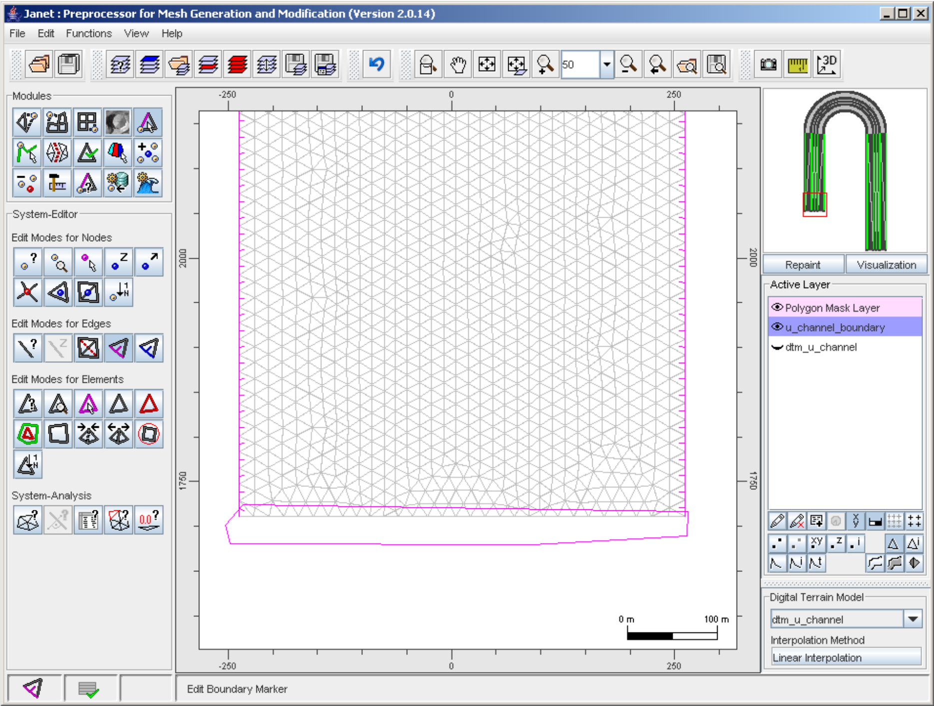

Figure 2-13. Deleted markers at the upstream boundary

2.6 Exporting the Mesh

The finite element mesh can be stored in various file formats (Janet Binary Format, Ticad-Ascii-Format, Telemac-Selafin, AVS/Express UCD, etc.). Some of these file formats simply store nodes and element connectivities, so the saved files do not contain the edited land boundary markers and integrated polylines!
For the specific example we use the “Janet Binary Format”, which stores all the described features to a file. The export is done with the [Save Layer As] function (file name: “u_channel_fe_mesh.jbf”).

Starting the HN-Model Martin

3

3.1 Loading the Finite Element Mesh

The finite element mesh created in the previous chapter is imported in the preprocessor with the [Load Layer] function as shown in the first chapter of this tutorial. It is important to adjust the setting of the selected item in the section “Reference System” of the model definition dialog according to the nodal z values being saved as depth or height values!
The finite element model Martin requires depth values at all mesh nodes, so if the existent mesh contains heights and the specific item is selected, all z values will automatically be converted when the simulation is started.

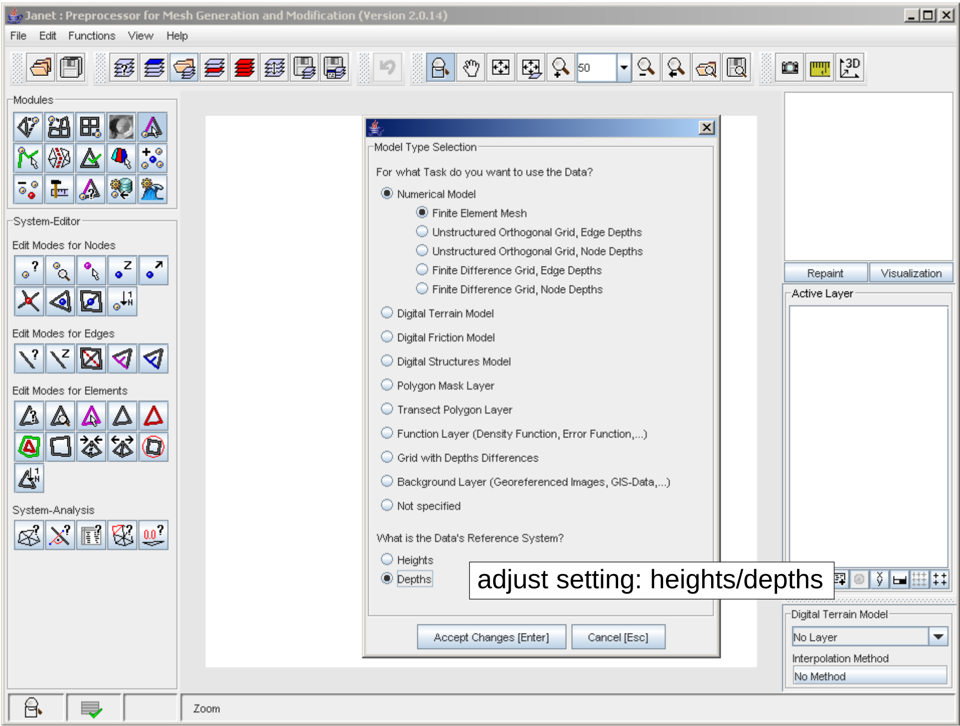

Figure 3-1. Model type definition

3.2 Generating Boundary Conditions

The preprocessor offers an user interface for editing boundary conditions. The function [Edit Boundary Conditions] can be found in the module “FE HN-Models Martin/Marina”. A boundary condition for a section at the open boundary is modified by selecting a single node of the specific boundary line. All input is then done in the dialog window of the boundary condition editor. For the given example we choose a water level boundary condition at the up- and downstream boundary and apply water level time series as presented in the next figures.

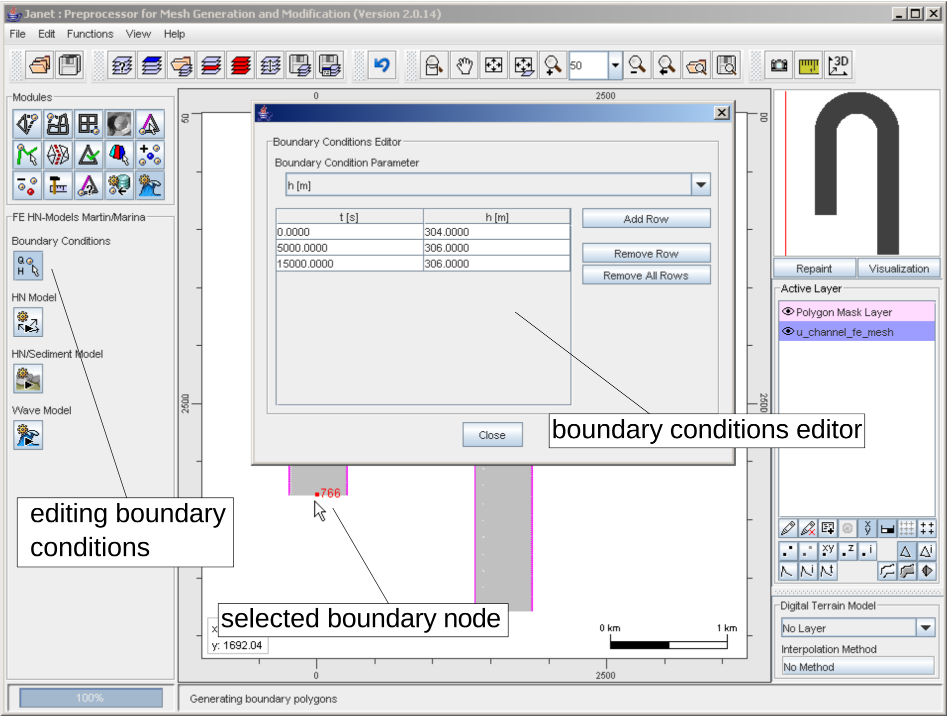

Figure 3-2. Boundary conditions at the upstream boundary

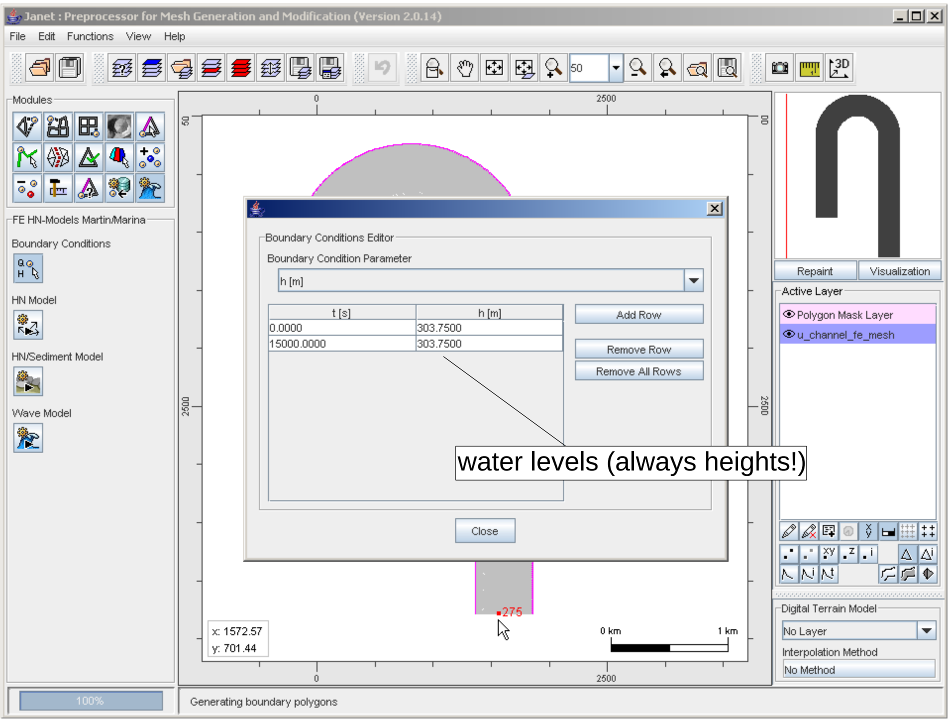

Figure 3-3. Boundary conditions at the downstream boundary

3.3 Starting the Simulation

A hydrodynamic simulation can be started with the [Start HN Model] function. Relevant parameters for the simulation are therefore modified in the dialog window. When the input is confirmed, the input data is veryfied, all files required for the simulation are written to the choosen project directory and finally the computation is started.

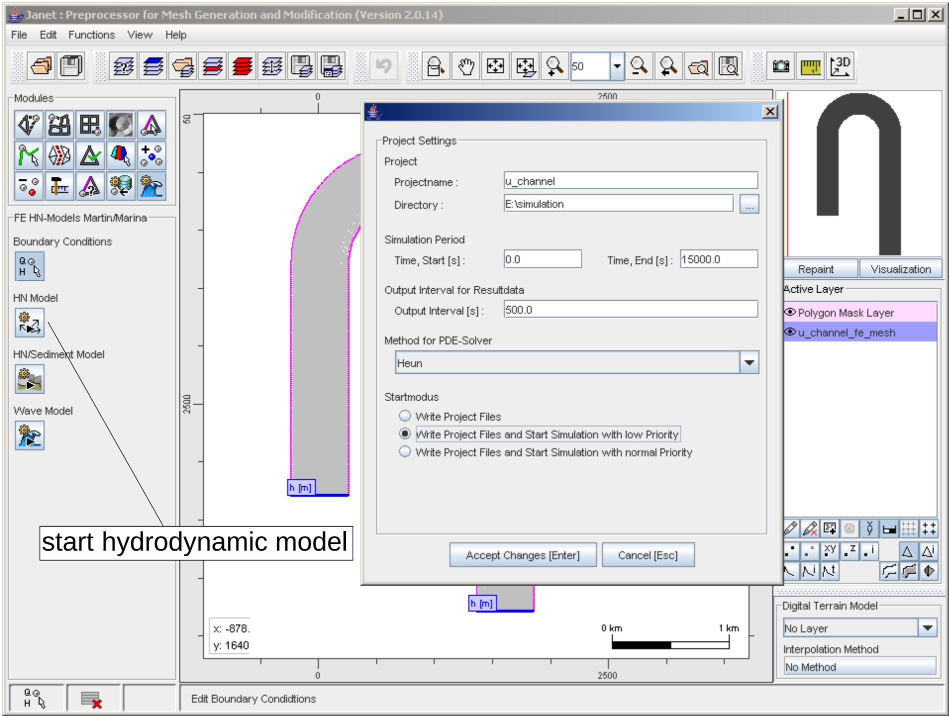

Figure 3-4. Modified simulation parameters for the given example

The simulation is started as a background process. The progress can be observed in the opened command line shell.

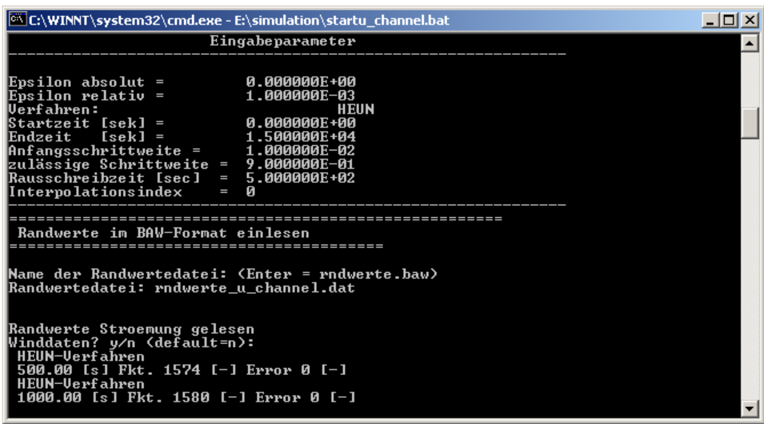

Figure 3-5. Observation of the simulation progress

Figure 3-6 presents screenshots of the simulation results. All images are generated with the postprocessing software Davit.

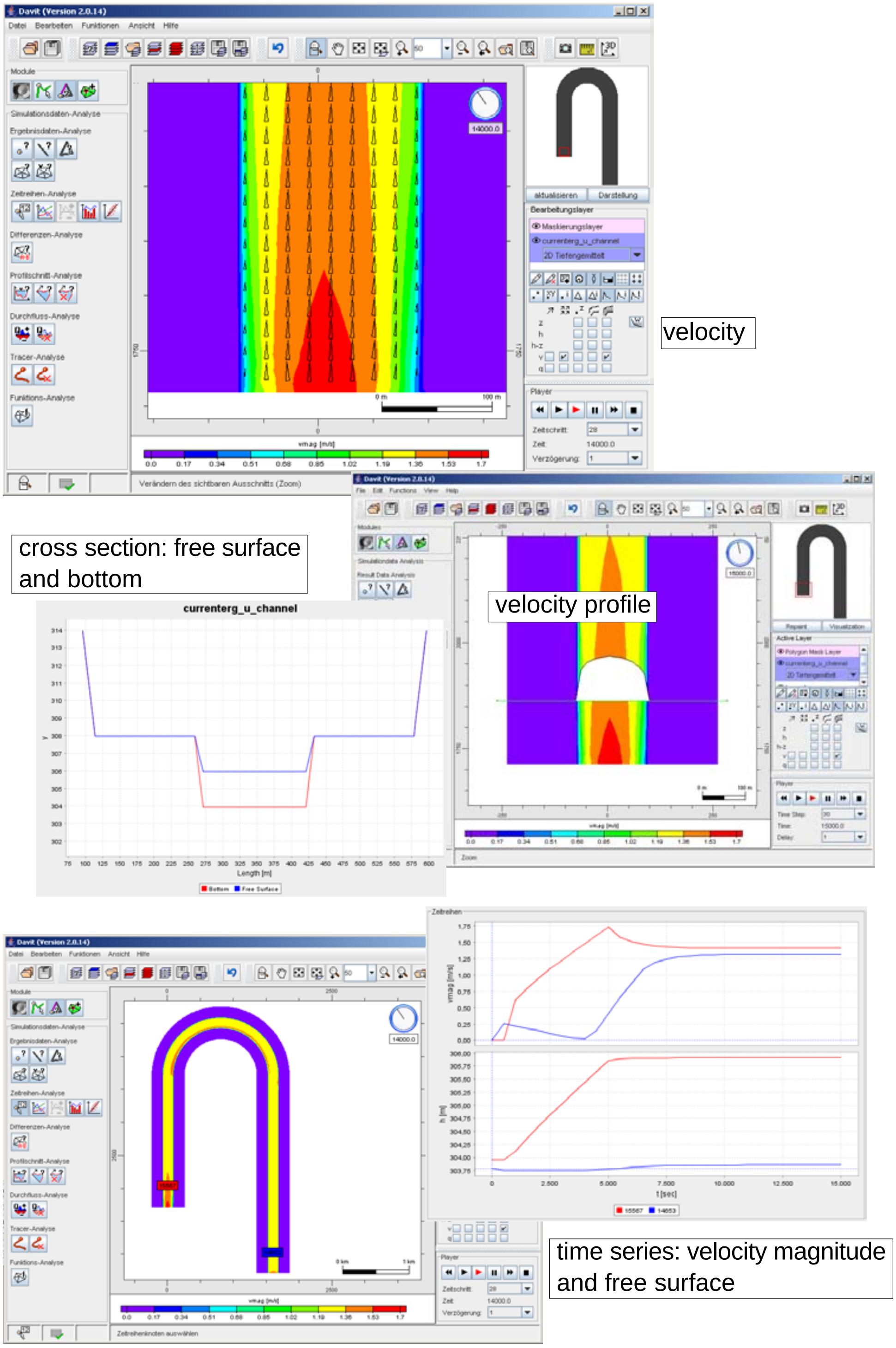

Figure 3-6. Simulation results (created with the postprocessor Davit)