Preprocessor Janet

First Steps

Telemac-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.

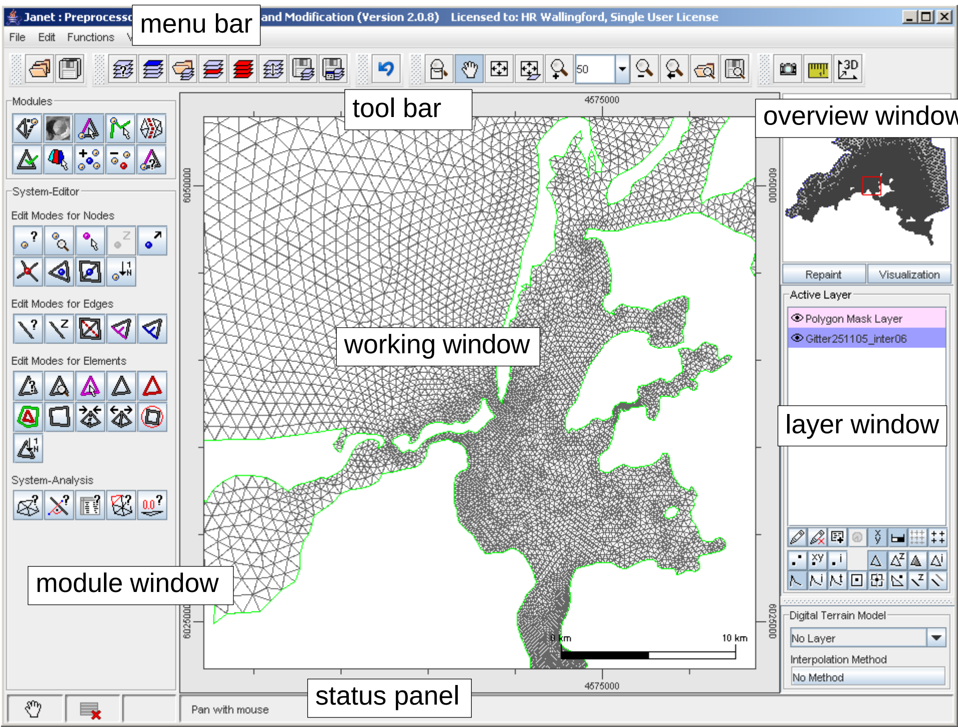

Figure 1-1. The graphical user interface

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 Telemac Mesh

A mesh in the selafin file format 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.
The specific file (e.g. “u_channel_be.sel” from the directory “examples”) is choosen in a file selection dialog. The import interface distinguishes between different byte orders, so binary files created with big- and little-endian byte order are supported by the preprocessor. For each type a file filter (“Telemac Binary Format, big-endian”, “Telemac Binary Format, little-endian”) is offered and has to be explicitely choosen. The example file “u_channel_be.sel” was stored with big-endian order and must be opened with the related filter.

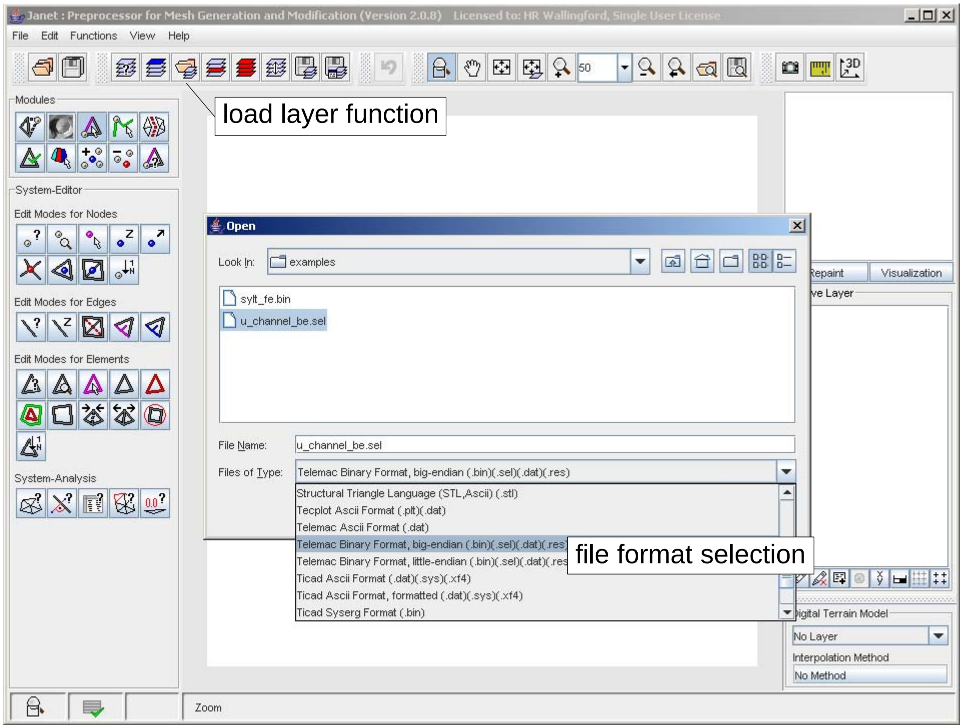

Figure 1-2. Loading the file “u_channel_be.sel”

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.

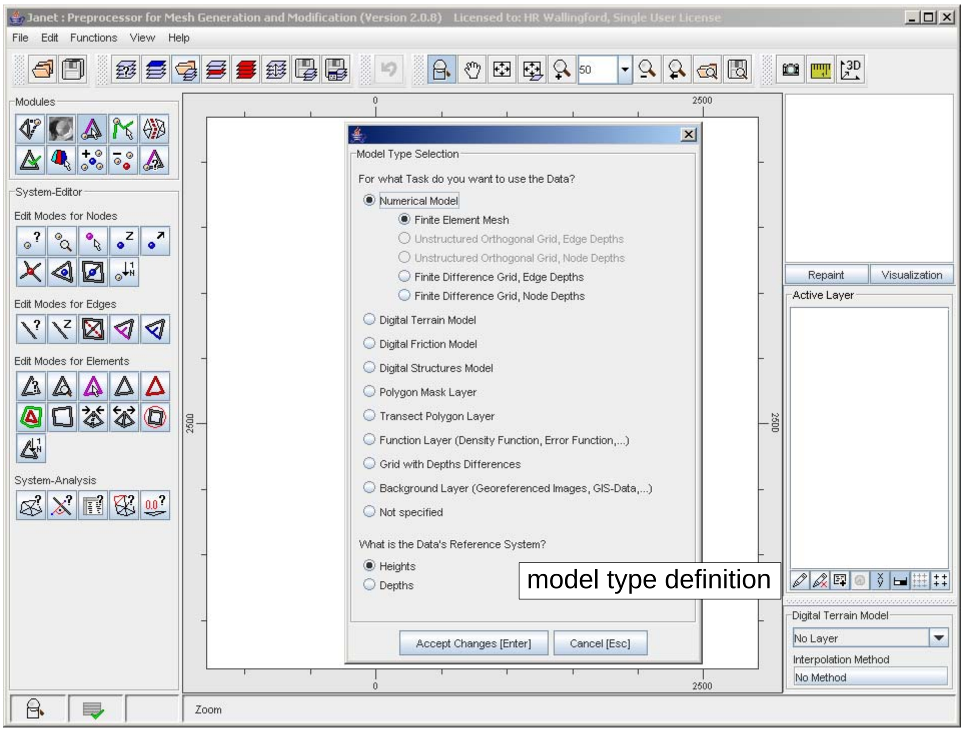

Figure 1-3. Model type definition

The file import is confirmed in a dialog window which lists all parameters and the number of time steps found in the selafin file.

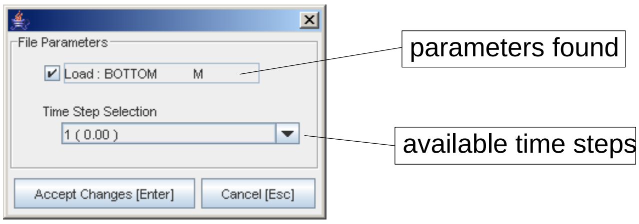

Figure 1-4. Parameter and time step selection

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.

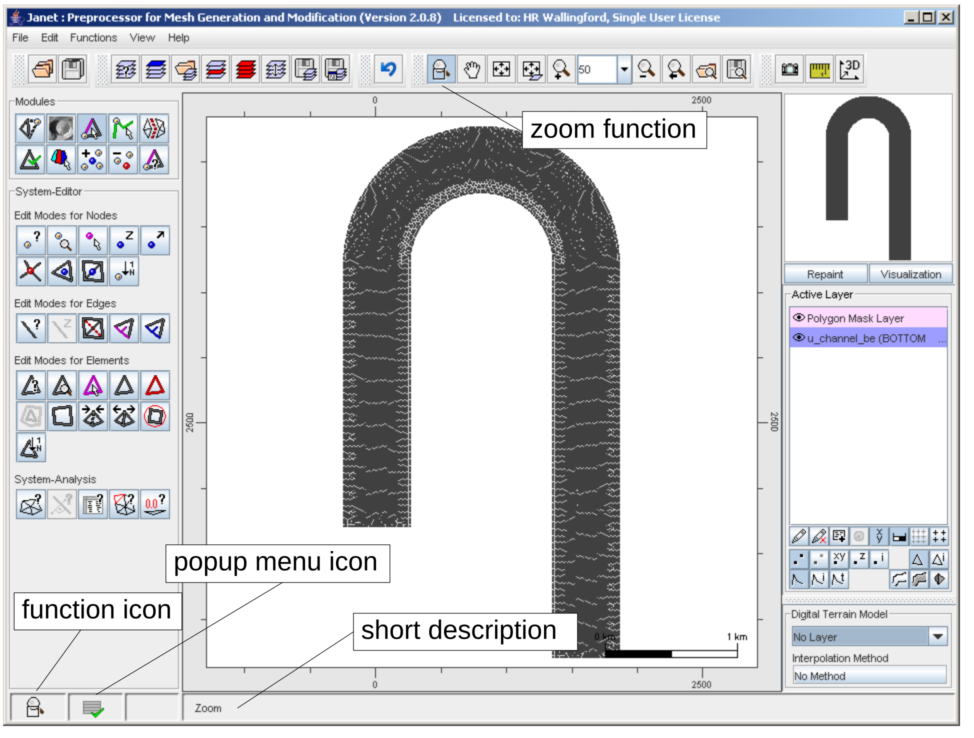

Figure 1-5. Main window after the file import

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.

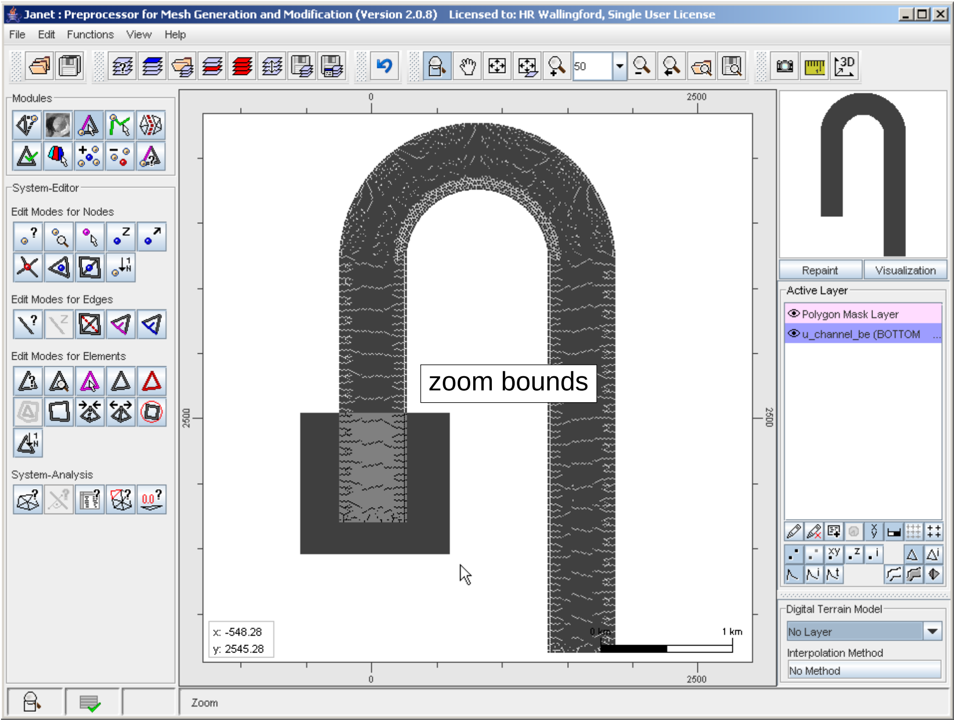

Figure 1-6. Changing the zoom state

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

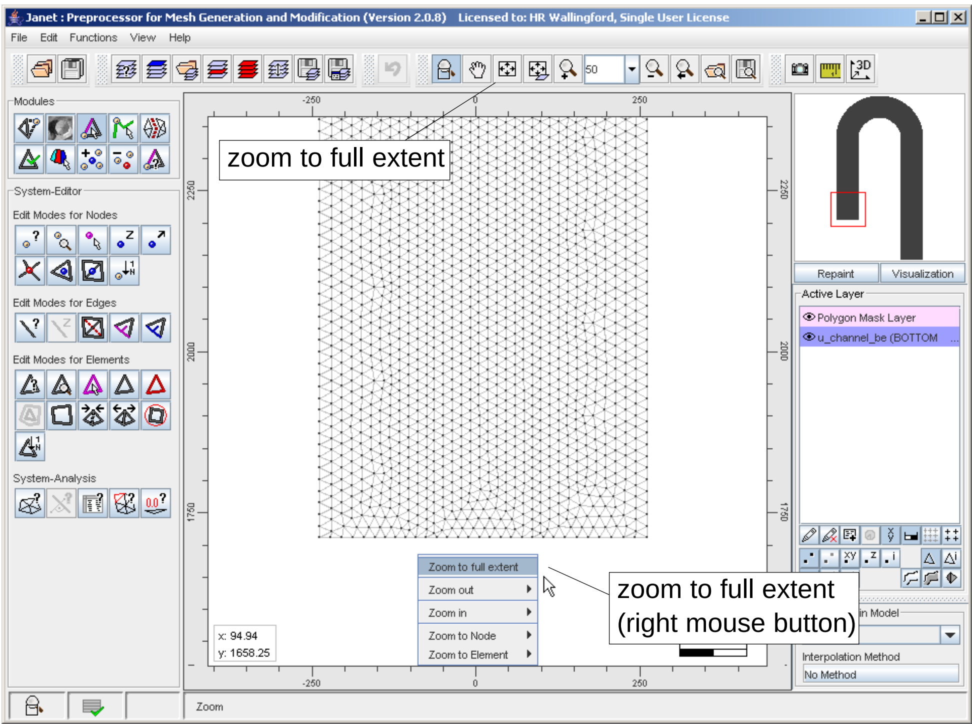

Figure 1-7. Zoom to full extent

To restore the default zoom state, either the menu item “Zoom to full extend” in the popup menu of the zoom function or the function button from the tool bar has to be selected (see Figure 1-7).

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.

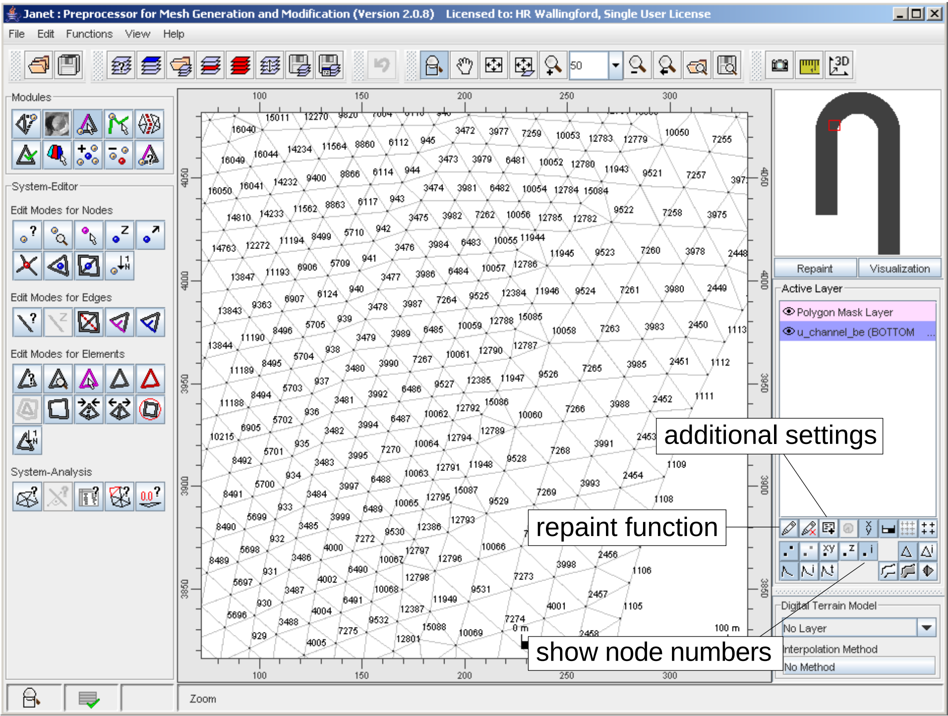

Figure 1-8. Modify visualization settings with quick edit buttons

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].

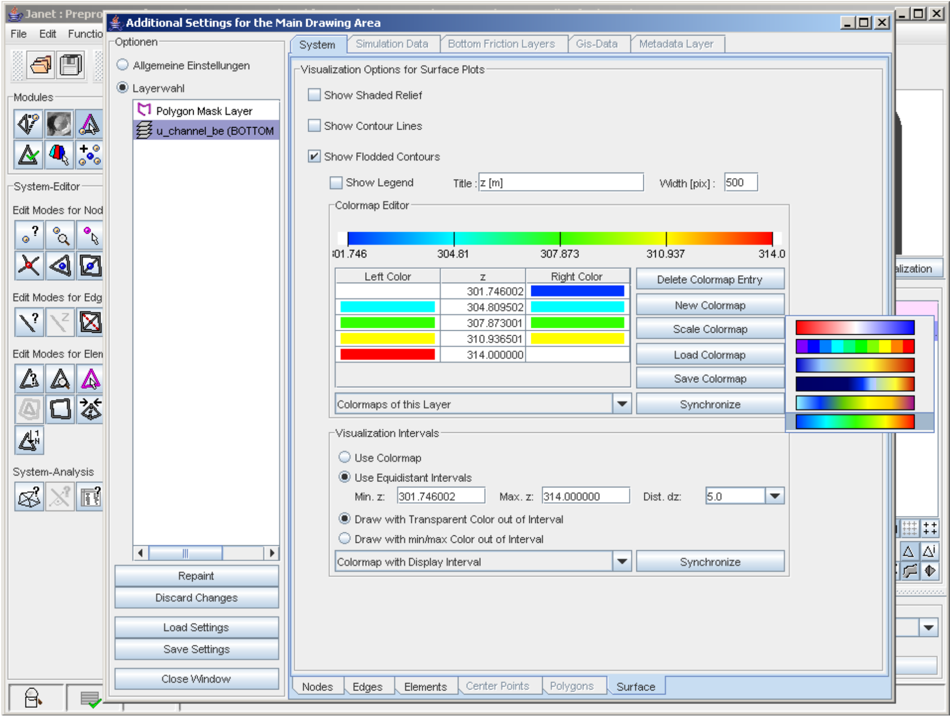

Figure 1-9. Show a surface representation of the grid

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_be.sel” leads to the result presented in the next figure.

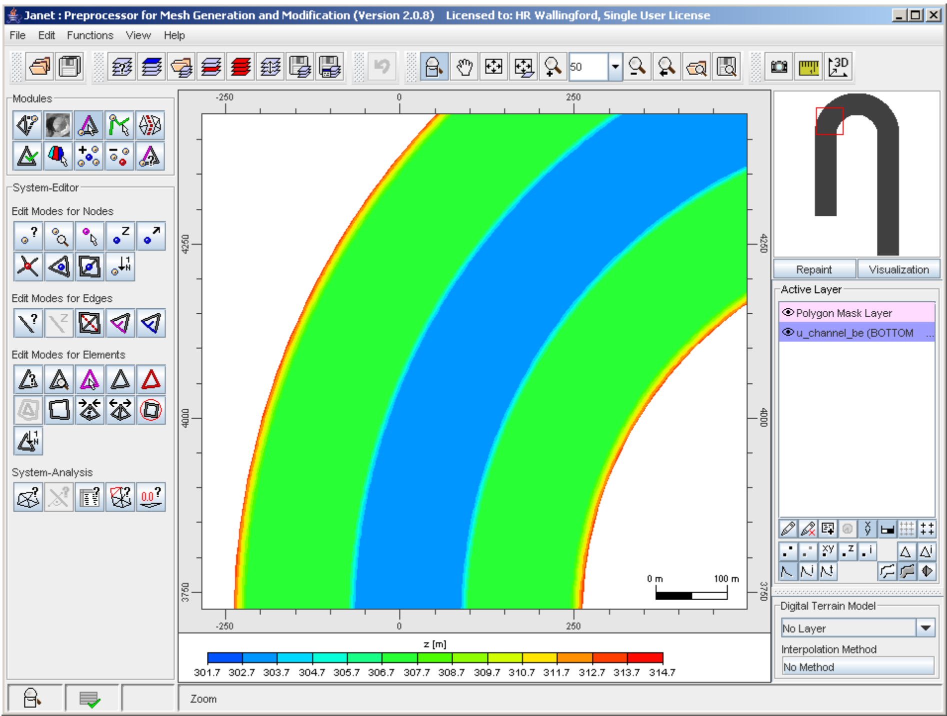

Figure 1-10. Surface representation of the grid

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.

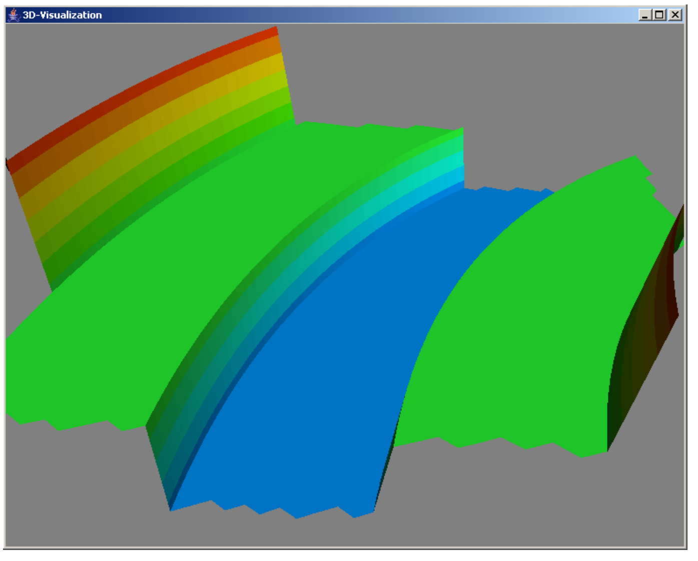

Figure 1-11. 3D viewer

Creating a simple Telemac Mesh

2

2.1 Basic Steps

In this chapter basic steps to create a first finite element mesh for Telemac are presented. The files “u_channel_boundary.jbf”, “dtm_u_channel_heights.jbf” and “u_channel_bottom_friction.gen” 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,
- importing a digital bottom friction model and finally
- exporting the model to the selafin file format.

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.

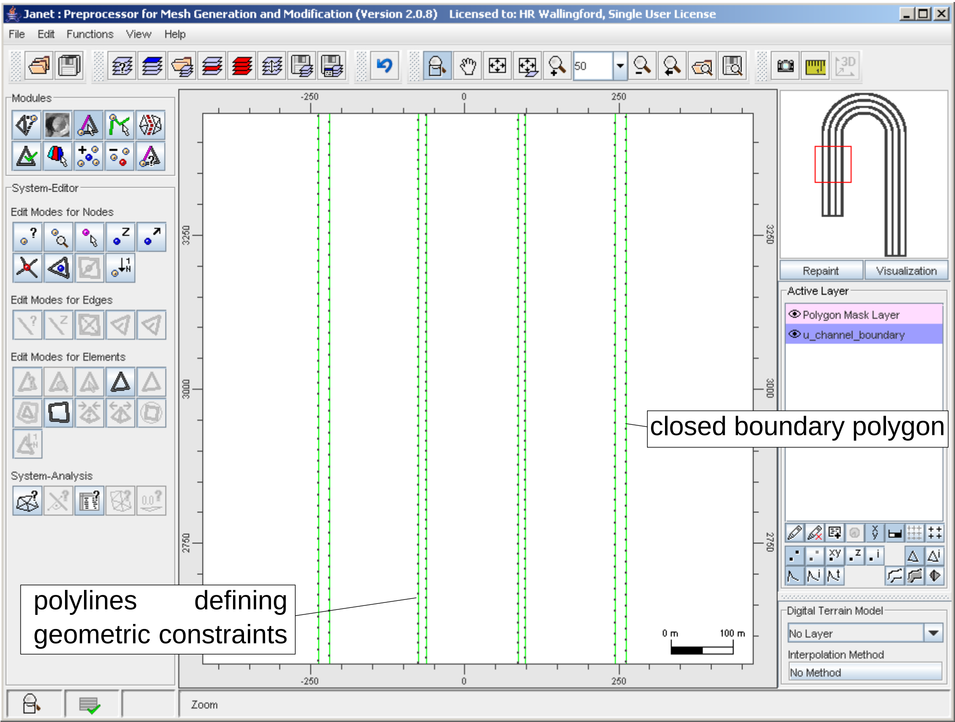

Figure 2-1. Boundary polygon and additional polylines

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.

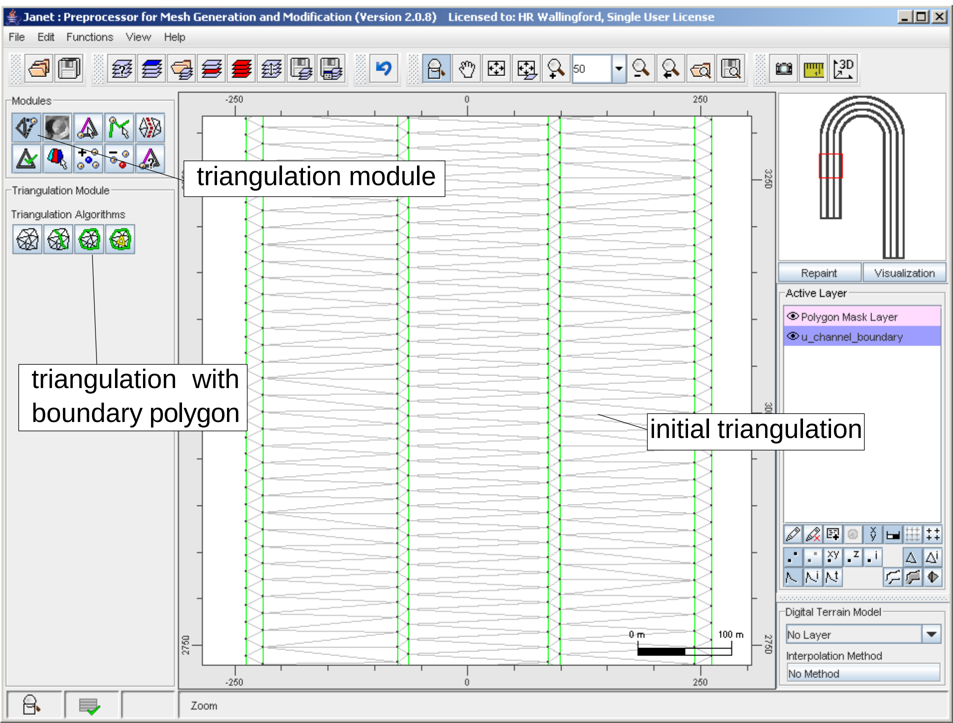

Figure 2-2. Boundary polygon and additional polylines

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 rastered nodes)

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

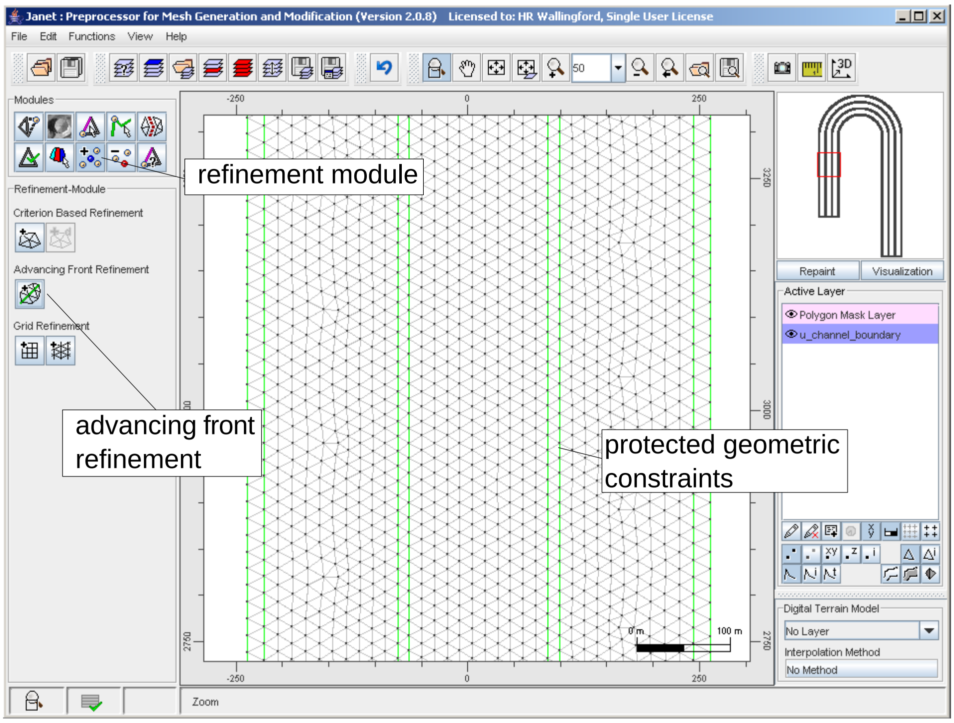

Figure 2-3. Boundary polygon and additional polylines

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_heights.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.

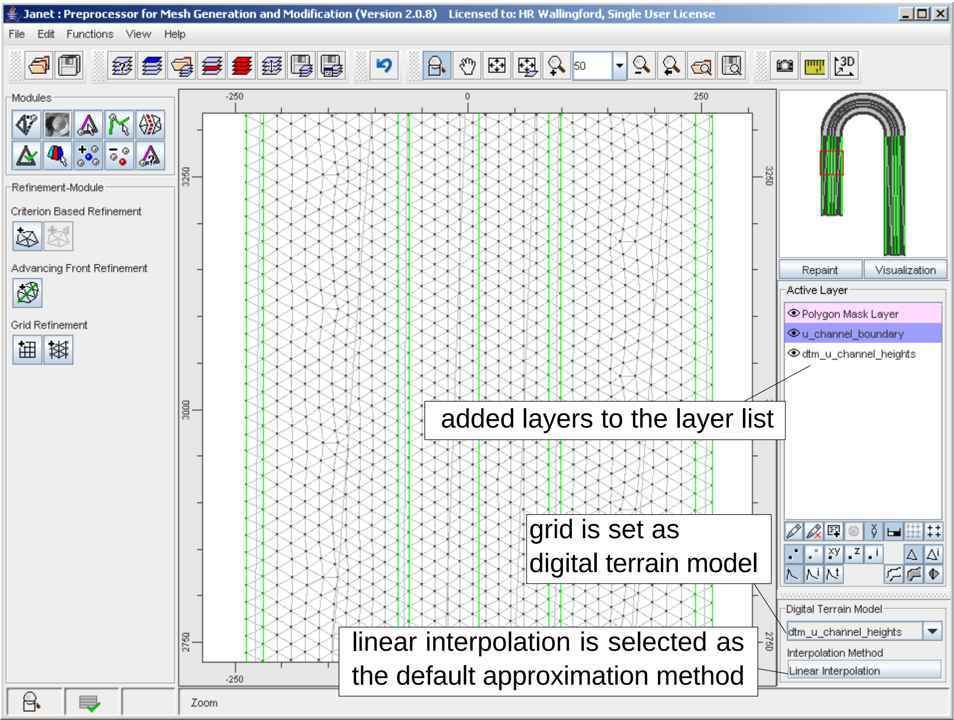

Figure 2-4. The imported digital terrain model

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.

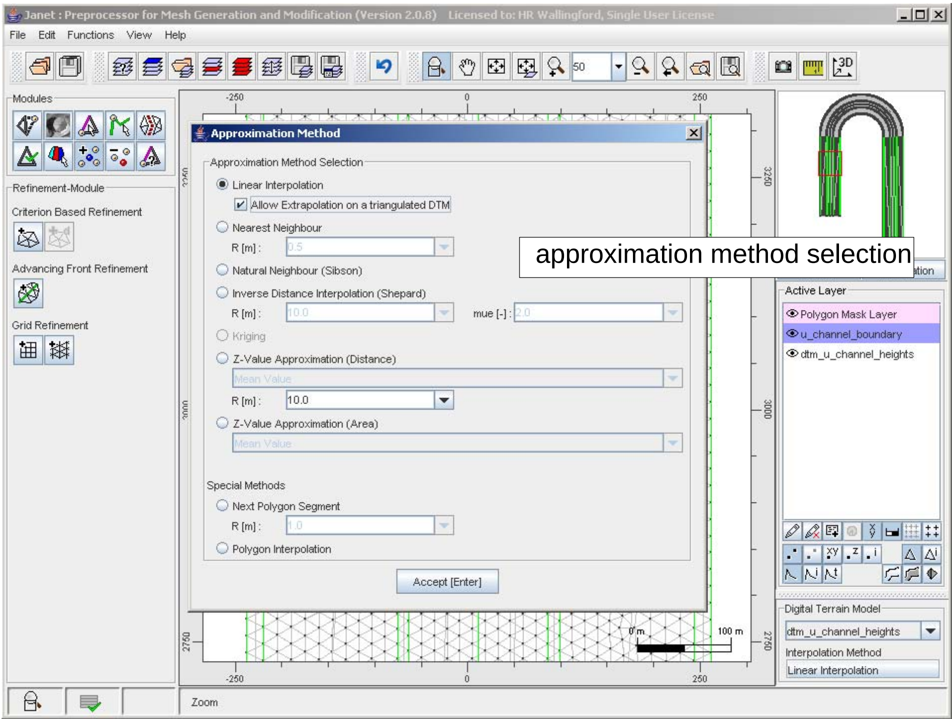

Figure 2-5. Editing the approximation method

In a next step the height 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".

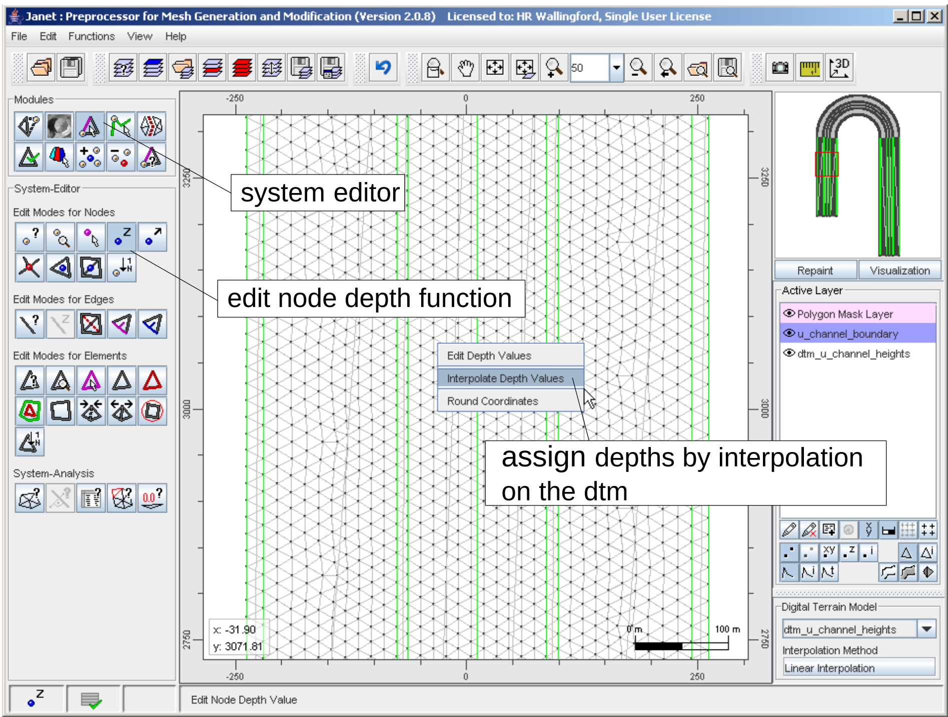

Figure 2-6. Starting the interpolation

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.

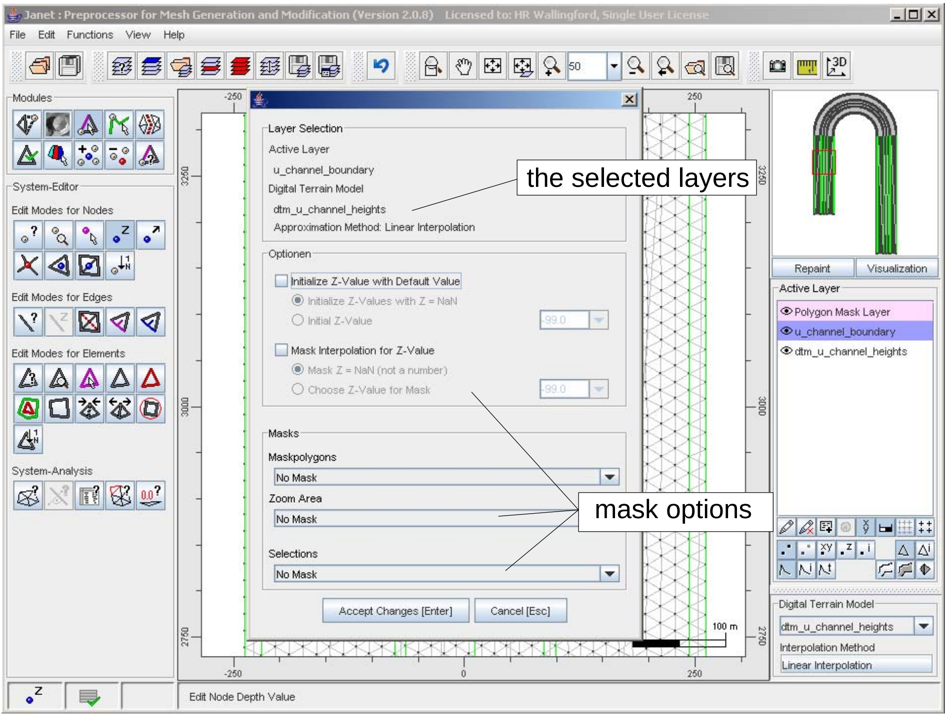

Figure 2-7. Dialog window to modify the options for the interpolation

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.

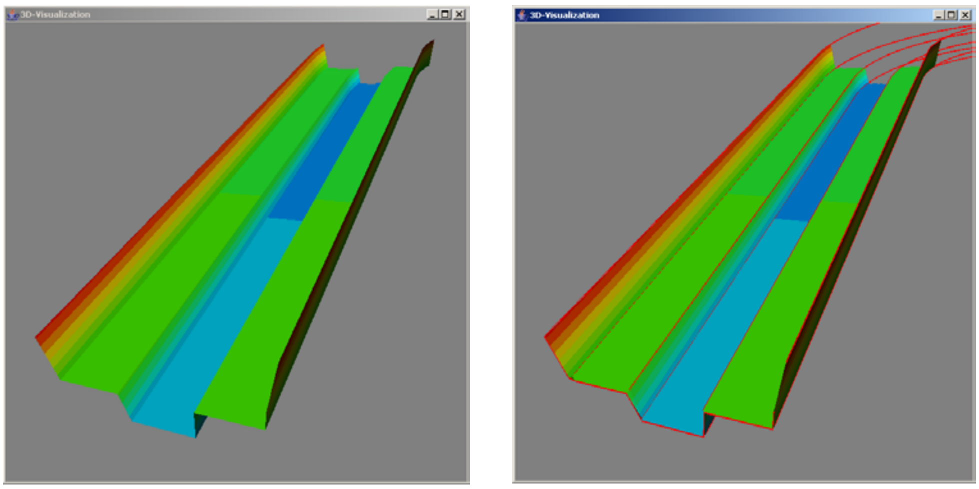

Figure 2-8. 3D view of DTM (left) and grid with interpolated depths (right)

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 Import a Digital Bottom Friction Model

The import of a digital bottom friction model prepares the export of the computational grid to the selafin file format.
This example illustrates the usage of polygonal bounded bottom friction layers. A single bottom friction layer is a set of polygons that represents a constant friction value. The bottom friction model consists of different layers which cover the entire domain.
The model “u_channel_bottom_friction.gen” is imported with the [Load Bottom Friction Layer Model] function from the bottom friction editor module.

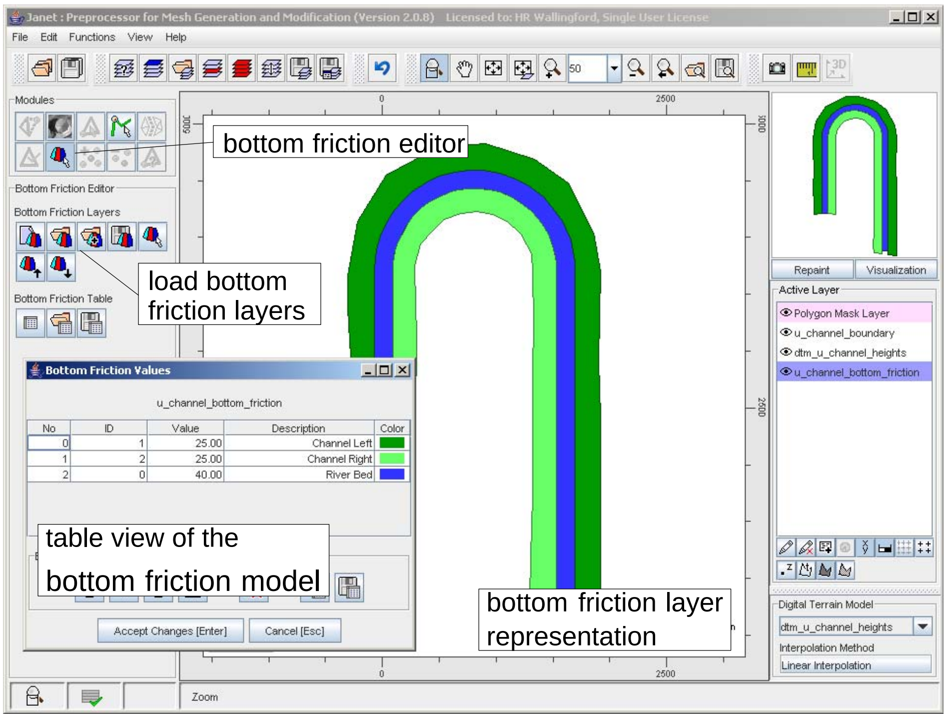

Figure 2-9. Imported bottom friction model

The imported bottom friction layers are presented with their assigned colors in the working window. Furthermore, the table view function offers an overview of all layers with the defined bottom friction values and related colors.

2.6 Masking Boundary Nodes

The special Telemac support offered by the preprocessor contains an integrated editor for the boundary condition file. To make the editing of the boundary condition file easier, user defined masks help to identify boundary sections in the table view of the editor.
Polygon masks can be defined on a special layer. The areas described by the closed polygons are used to mask all nodes that are enclosed by these polygons. To active this layer, the entry “Polygon Mask Layer” in the layer list has to be selected. Polygons enclosing the in- and outflow boundary sections

are then created with the [Define Polygon] function in the polygon editor. A description is finally assigned to each polygon (e.g. ”inflow boundary”) with the “Change identifier of a polygon” function.
Figure 2-10 shows one of the two generated polygons for the example.

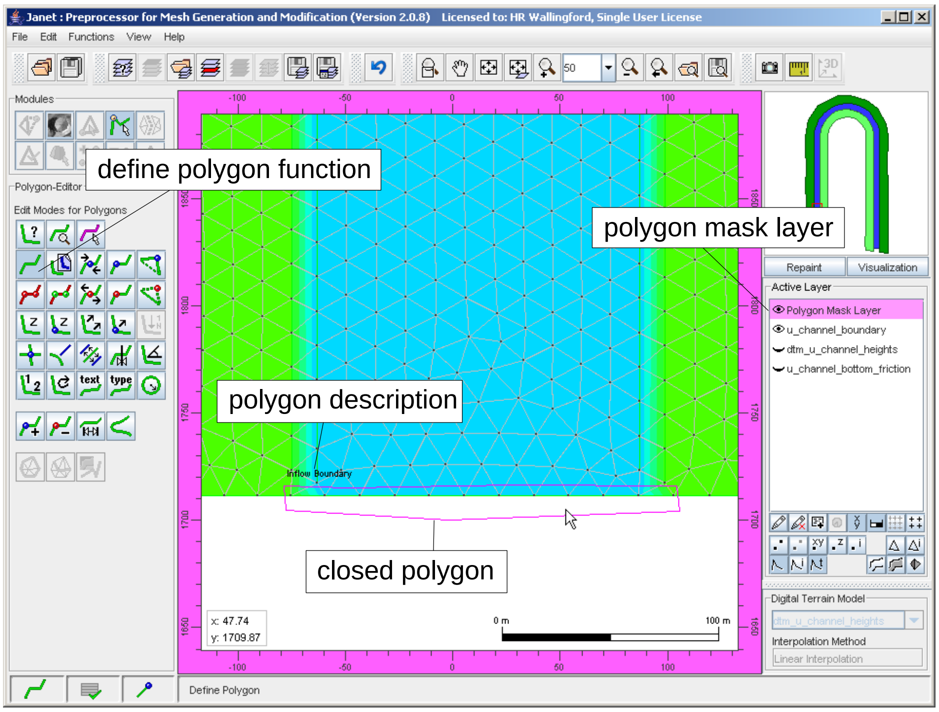

Figure 2-10. Definition of a polygon mask

2.7 Export to the Selafin File Format

The modified model is finally exported to the selafin file format with the [Save Layer as] function and choosing the file filter “Telemac Binary Format”.
The export is done in several steps. In the first step, the mesh is checked for elements that contain boundary nodes only. If any elements are found, the element numbers will be listed in a seperate dialog window. The user is offered an option to have the specific elements automatically removed (by insertion of additional nodes).
The next step opens the boundary condition file editor (see Figure 2-11). Single boundary condition values as well as boundary condition sections masked with polygons can be comfortably edited.

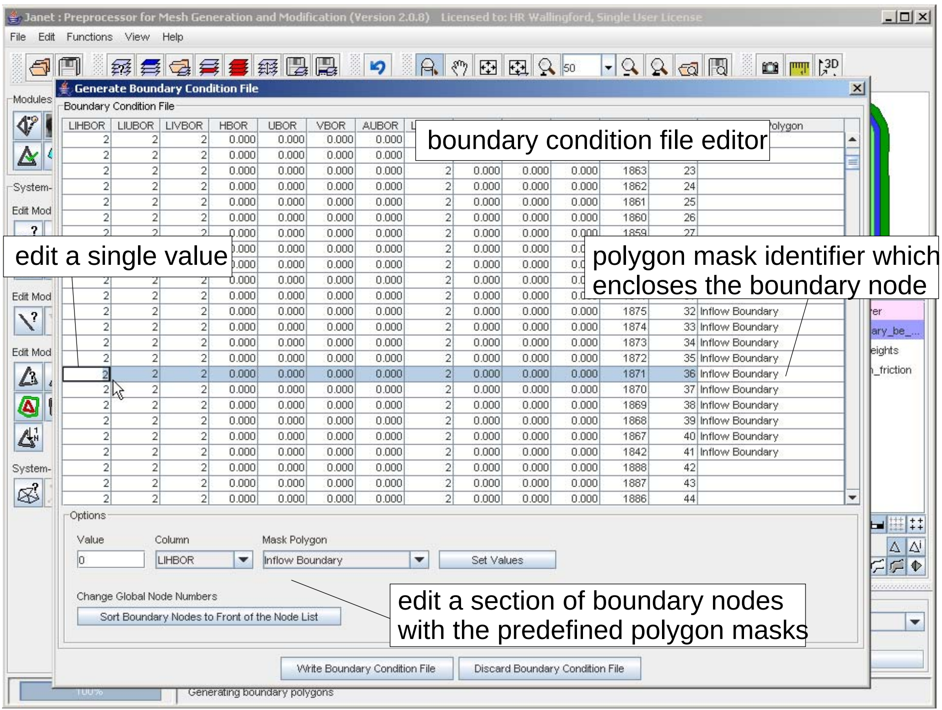

Figure 2-11. Boundary condition file editor

Finally the export parameters are defined. For the current example a selafin file with the parameters “BOTTOM” and “BOTTOM FRICTION” is created. The specific models of the layer list are selected from the listboxes and the export is confirmed with the [Accept Changes Button].

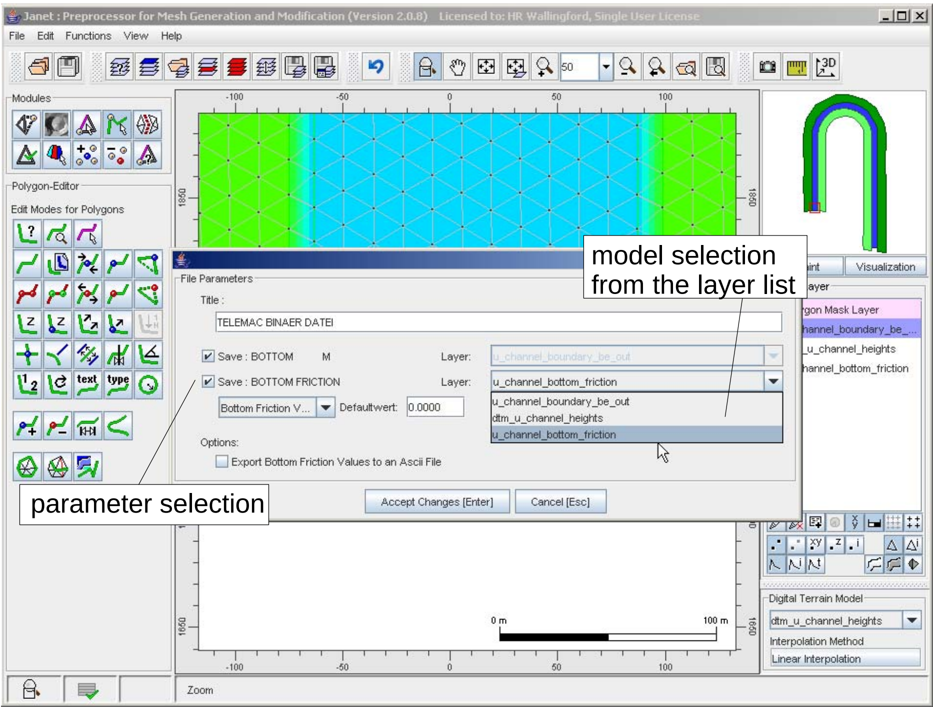

Figure 2-12. Output parameter selection