Groundwater flow and piping calculation module

D-Geo Flow

Deltares systems

Finite Element Model DGFlow in Delta Shell

Deltares

Enabling Delta Life

Manual

D-Geo Flow

D-Geo Flow in Delta Shell

User Manual

Version: 1.0.0
Revision: 51382

5 September 2017

D-Geo Flow, User Manual

Published and printed by:

Deltares		telephone:	+31 88 335 82 73
Boussinesqweg 1		fax:	+31 88 335 85 82
2629 HV Delft		e-mail:	info@deltares.nl
P.O. 177		www:	https://www.deltares.nl
2600 MH Delft			
The Netherlands			

For sales contact:

telephone: +31 88 335 81 88
fax: +31 88 335 81 11
e-mail: sales@deltaressystems.nl
www: http://www.deltaressystems.nl

For support contact:

telephone: +31 88 335 81 00
fax: +31 88 335 81 11
e-mail: support@deltaressystems.nl
www: http://www.deltaressystems.nl

Contents

List of Figures

List of Tables

1 Installing Instructions

1.1 Installing D-Geo Flow as Delta Shell plugin

1.1.1 Installing D-Geo Flow

After downloading the D-GeoFlow(*version number*).msi, this file should be double-clicked in order to start the installation wizard. The install wizard will take you through the installation process including the creation of program shortcuts. Please select the 'Standard' installation profile when prompted. After installing DS-FLEX the *.lic file needs to be registered. Once done, D-Geo Flow is ready to be used.

1.2 Starting D-Geo Flow

1.2.1 Starting D-Geo Flow for the first time

As mentioned in the previous section, the installation wizard will create a program shortcut in both the Windows Start Menu as well as on the desktop. Please select one of the two to launch D-Geo Flow.

2 A Guide to this manual

2.1 Introduction

This User Manual concerns the (un)saturated groundwater flow and piping module D-Geo Flow.

2.2 Overview

To make this manual more accessible we will briefly describe the contents of each chapter.

This manual consists of the following chapters. In Chapter 3: 'Module D-Geo Flow: an overview' on page 5 the D-Geo Flow module is explained on a general level. In Chapter 5: 'Module D-Geo Flow: Getting started' on page 23 is explained how to get started with the D-Geo Flow module.

D-Geo Flow contains the following Key Features:

⋄ 2D transient saturated and unsaturated groundwater flow calculation

⋄ Sellmeijer model implemented for predicting pipe progression

⋄ Complex subsurface/multi-layer configuration definition through user interface

⋄ Application of time dependent hydraulic loads

⋄ Graphical and numerical presentation of output of groundwater flow and pipe development over time

2.3 Versions and Revisions

This is version 1.0.0 of the User Manual.

2.4 Known issues and recommended settings

In this version of D-Geo Flow Deltares acknowledges the following known issues:

⋄ In the *drawing mode*: sometimes, when due to overlapping geometric objects (e.g. soil layers) the intersections results into very small, sharp-edged triangles, the meshing of these triangles will fail or are not meshed properly.

⋄ In the *drawing mode*: moving of individual points by using the mouse is disabled.

For this version of D-Geo Flow Deltares recommends the following settings:

⋄ To use boreholes for generating (sub)soil geometries.

⋄ To set the decimal symbol to a 'point'.

3 Module D-Geo Flow: an overview

3.1 What is D-Geo Flow

Software module for groundwater flow simulations and for modeling piping under dikes and dams. D-Geo Flow uses the Delta Shell User Interface in combination with the Finite Element Model DgFlow which includes a piping module.

3.2 System Requirements

Processor Intel i5 Dual Core 2 GHz

RAM Minimal 1 GB

Disk space 500 MB (excluding model and project files)

Screen resolution 1280 x 1024, 64 bit color depth (recommended)

3.3 The Delta Shell framework

D-Geo Flow is a Delta Shell plugin. For more information about Delta Shell, see next chapter.

4 Delta Shell Basics

4.1 Windows

As Delta Shell is an integrated modelling suite, the application is project-based. Within a project several models may be run and combined.

The main user interface is organized in a set of tool and document windows. An example is given in Figure 4.1. The tool windows show properties of the current project, whereas document windows are used to visualize or edit a specific data type. Tool windows can be docked where you prefer — even at a second display. Document windows are, when placed within the framework, always in the central area but may also be docked stand-alone (on a second display, for example). Examples of tool windows are:

- ◇ **Project**
- ◇ **Map**
- ◇ **Region**
- ◇ **Properties**
- ◇ **Chart**
- ◇ **Data**
- ◇ **Time navigator**
- ◇ **Messages**
- ◇ **Toolbox**

Examples of document windows are:

- ◇ Map(s)
- ◇ Editor(s)
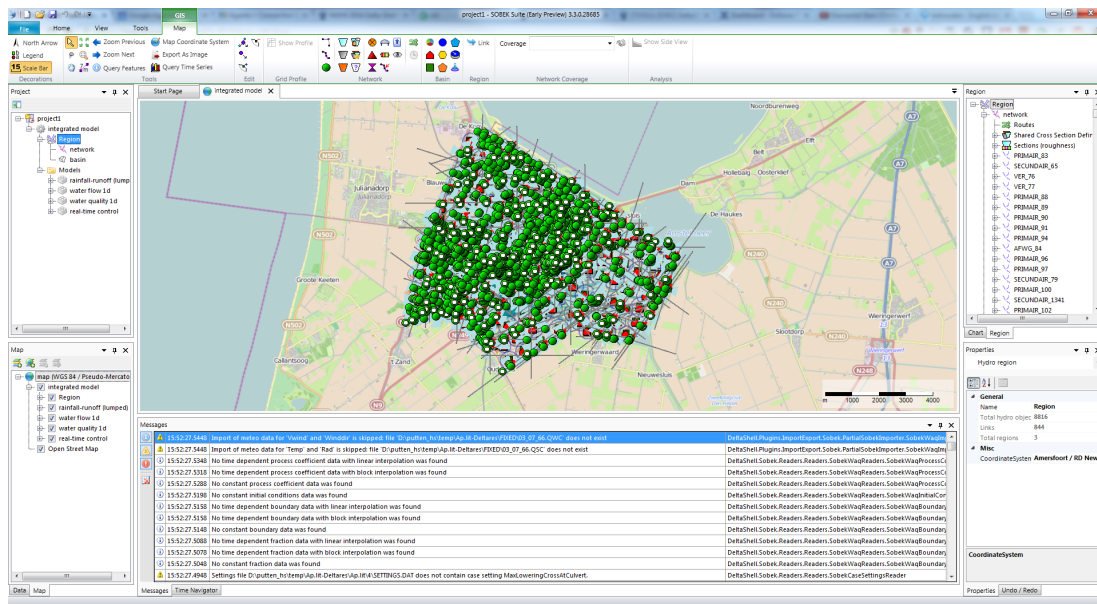
Figure 4.1: *Overview of the graphical user interface, example for a SOBEK 3 model.*

In this chapter all tool windows, menus, dockable views, context menus, and ribbons and toolbars will be described. Map functionality and the spatial editor are treated in separate chapters. The specific editors for the different models are described in the user manuals belonging to those model plug-ins.

4.1.1 Projects

The **Project** window is the main navigation window for the project data, showing the total workspace in a tree view (Figure 4.2). In the **Project** all project components are shown. For a project containing a one-dimensional flow model and a map these may look like:

◇ The model (in this case a 1-dimensional flow model)
    ▢ The model input
        ○ The network(s)
        ○ The computational grid(s)
        ○ The initial conditions
        ○ The boundary conditions
        ○ Lateral data
        ○ Roughness
        ○ Wind data
    ▢ The model output/results
◇ The map

All project items with sub-levels can be collapsed by a mouse-click on the '−' sign in the tree view. Project data can be sorted by adding new folders to the project tree view and moving models or movable items to designated folders. By clicking on the top left icon in Figure 4.2 the active item in the central Map is located in the tree view.
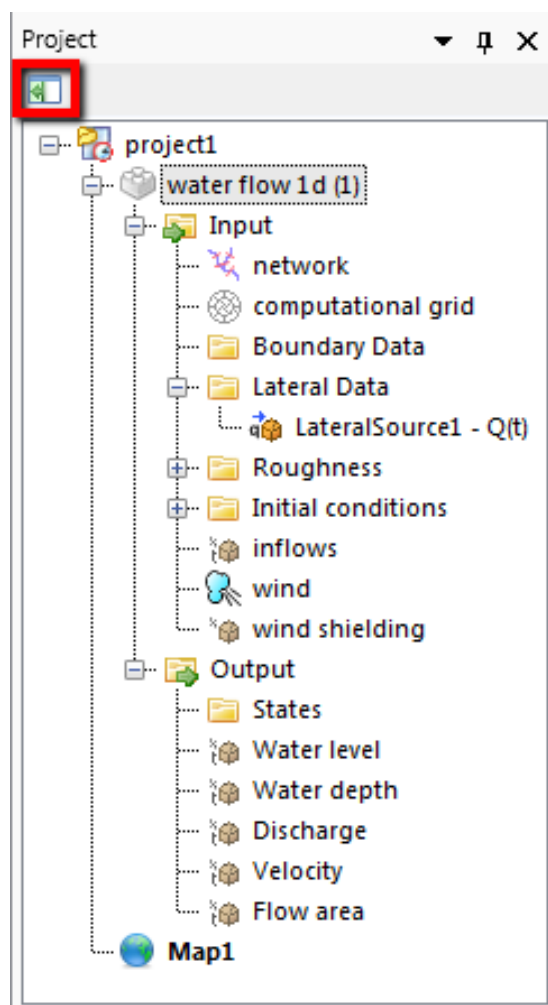
Figure 4.2: The project tree window.

Several possibilities exist to work with the tree view:

◇ Left mouse-click to select
◇ Right mouse-click gives a context menu with available actions
◇ Double-click to show a map or editor in the main (central) window, depending on the parameter

4.1.2 Main (central) window

The main window (Figure 4.3) is by default always placed in the middle of the screen. It can also be docked separately, for example on a second display. It is used to present a map for all geo-referenced modeldata, the editors for other data, and results in charts. The editors for other data are model-specific and therefore described in the manuals for the various model plug-ins.

All items with a geo-reference will be presented on the central map, for example: network, computational grid and output data as layers, comparable to a geographical information system (GIS).
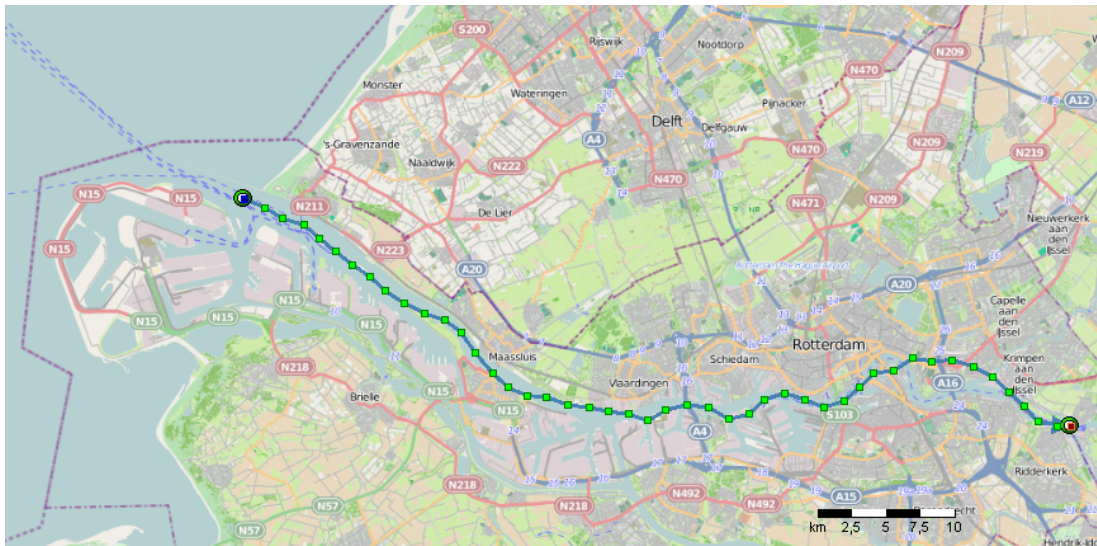
Figure 4.3: The central map view.

When working in the central map, for example on a network, it is possible to add, adjust or delete network components, which is described thoroughly in the D-Flow 1D manual.

Results in charts windows (Figure 4.7) are presented by combining a table view and a 'xy'-plot. The user can visualize separate data points by selecting rows in the table view. The data that is presented in the chart window may also be exported to a *.csv file by clicking the *Csv export* button.
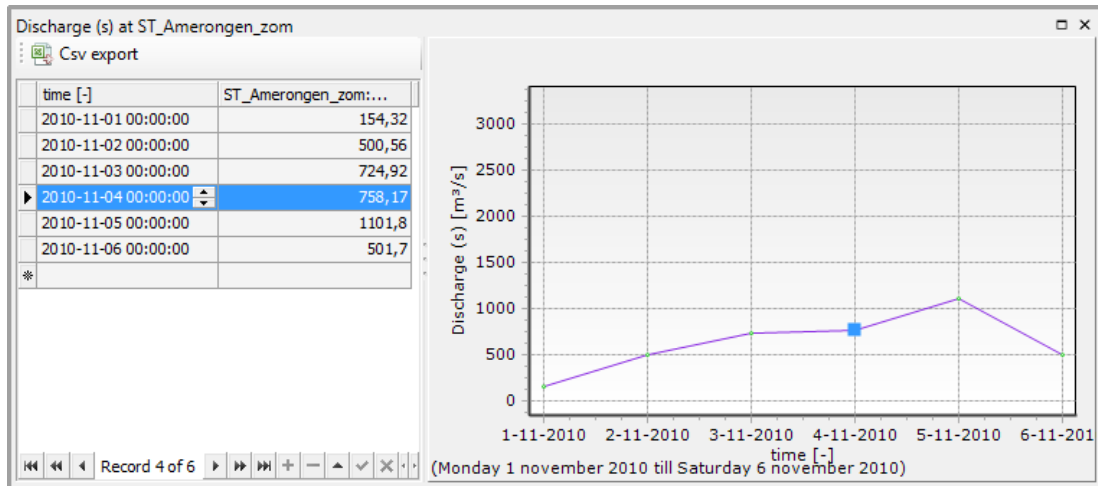
Figure 4.4: *The chart window view.*

4.1.3 Map

The **Map** window (Figure 4.5) manages the active map. In this window layers within the active map can be shown, hidden or adjusted. With the four icons in the top left of the window new <shp>- or <wms>-layers can be added, removed or exported. With the icons in the top right of the window, the window can be removed or hidden. The window can be retrieved by clicking on *Map* in *View* ribbon.
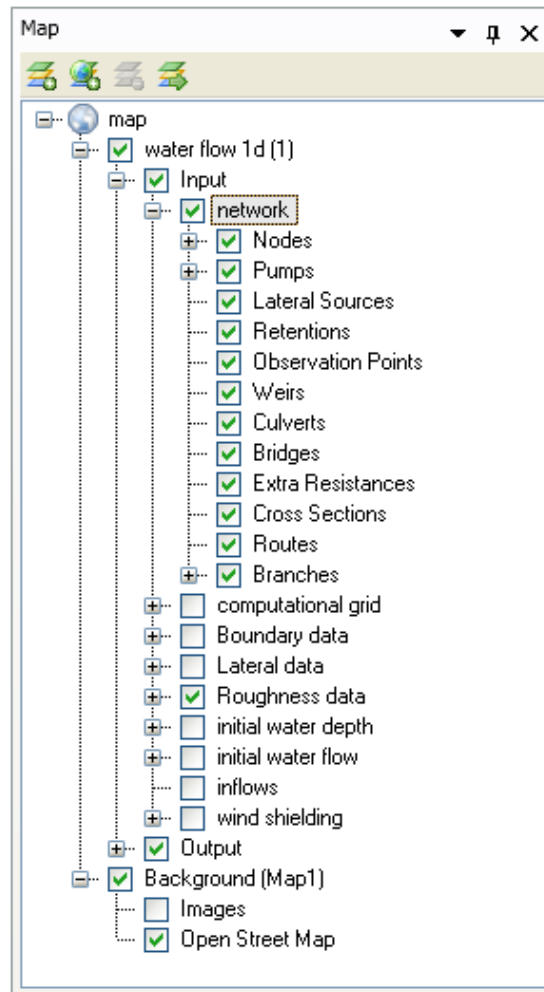
Figure 4.5: The map window.

4.1.4 Data

The **Data** window (Figure 4.6) can be used to inspect the contents of available data items, when selected in the project window.
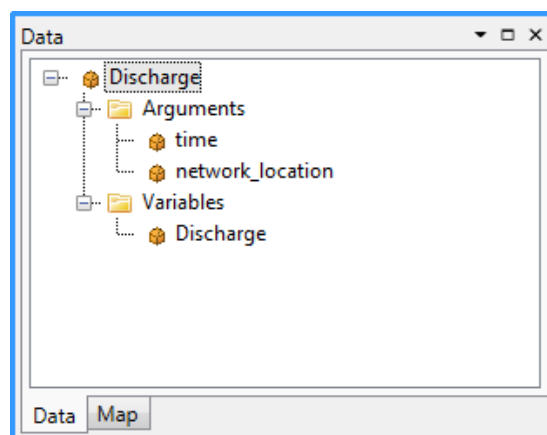
*Figure 4.6: The **Data** window.*

4.1.5 Chart

The **Chart** window (Figure 4.7) can be used to stack/unstack chart series of the same type or to convert chart series to a certain type (area, line, point, or bar series), Furthermore, series can be selected or deselected within this window.
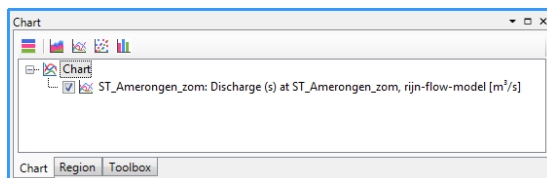
Figure 4.7: Example of the chart window.

4.1.6 Region

In the **Region** window (Figure 4.8) all hydrological objects are shown, like river-networks, -catchments or waterbodies. The **Region** window is related to the selected model. While in the central map the model objects are arranged according to their spatial location, the **Region** sorts the model objects by category and chainage. A component that is selected in the **Region** window is also automatically selected in the central map and vice versa.
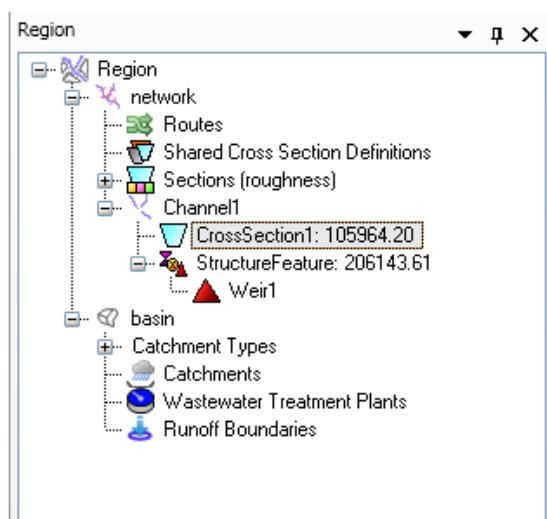
Figure 4.8: Example of the region window.

4.1.7 Toolbox

In the **Toolbox** window (Figure 4.9), users can quickly add a variety of items (models, stand-alone items) to the project tree, by selecting 'Add new item' or 'Add new model' after a right-mouse-click on the desired item. Furthermore, users may manage their Python scripts within the **Toolbox** window.
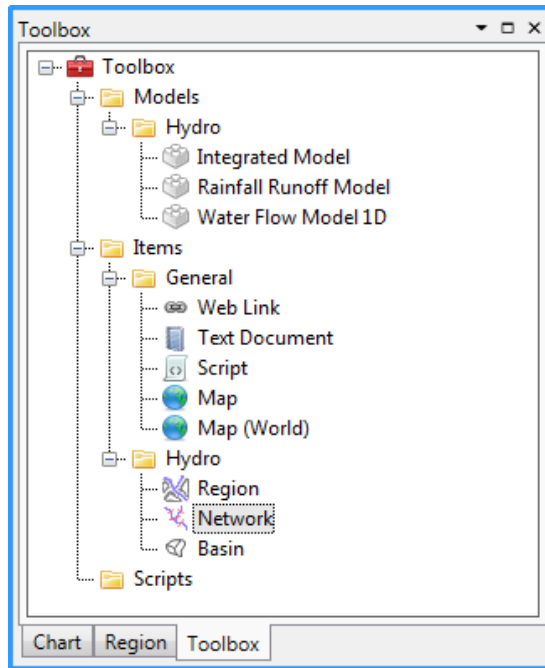
Figure 4.9: *Example of the **Toolbox** window.*

4.1.8 Properties

The **Properties** window shows properties for an active selection of the graphical user interface. When a model object is selected in the **Region** window it shows the properties of this object. Accordingly, the **Properties** window of an item selected in the **Project** shows data related to the selected item, for example the simulation time of a <flow model> or a list with output parameters when clicking on the <output> entry. Figure 4.10 shows an example for the properties of a flow model.

In the **Properties** window data can also be edited. If the property grid is insufficient to display the information, for example in case of time series, an additional editor can be opened.
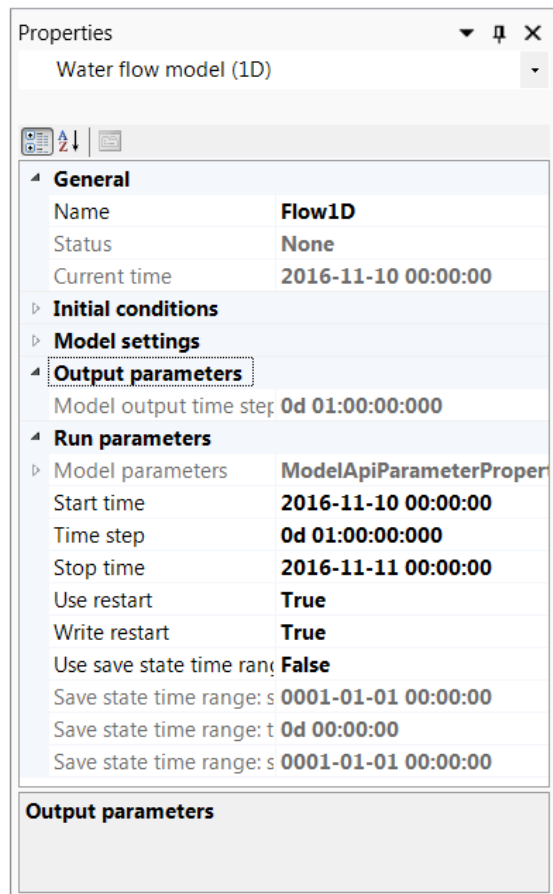
Figure 4.10: Example of a property grid in the properties window of a flow model.

4.1.9 Time navigator

The **Time navigator** is used to navigate through time(steps) of any time-dependent variable. An example is given in Figure 4.11. Each map or side view that shows time-dependent variables has its own time navigator. Users may also play results of time-dependent variables continuously.
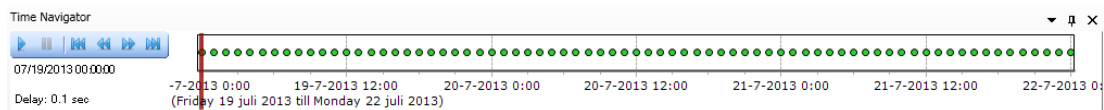
*Figure 4.11: Example of the **Time navigator** window.*

4.1.10 Messages

The **Messages** window (Figure 4.12) is a logging window. Messages sent from models or different parts of the system are shown here. When a message is too large to fit within the **Messages** window, the user can open a single message (Figure 4.13) separately by right-mouse-clicking the message and selecting the 'Show details' option.
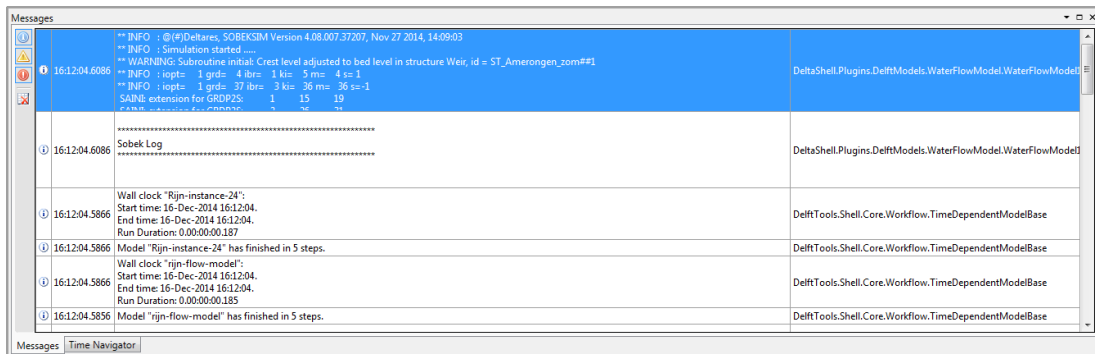
Figure 4.12: The messages window.
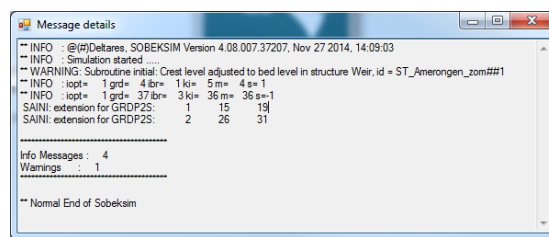
Figure 4.13: The message detail window.

Within the **Messages** window the user may select the verbosity of the shown messages, ranging from 'Info messages' to 'Warning messages' to 'Error messages'. It is also possible to clear all messages by clicking ▣ .

Furthermore, a run report is shown in the output in the **Project** for each model simulation. This run report contains all the messages (from Delta Shell and the model plug-ins) that occur during a simulation.

Finally, an application log is kept for each session of Delta Shell in the project database. In this log-file, which can be accessed through the *File/Help* or *Home* menus, all messages are stored.

4.2 Dockable views

The Delta Shell framework offers lots of freedom to customize dockable views, which are discussed in this section.

4.2.1 Docking tabs separately

Within the Delta Shell framework the user can dock the separate windows according to personal preferences. These preferences are then saved for future use of the framework. An example of such preferences is presented in Figure 4.14, where windows have been docked on two screens.
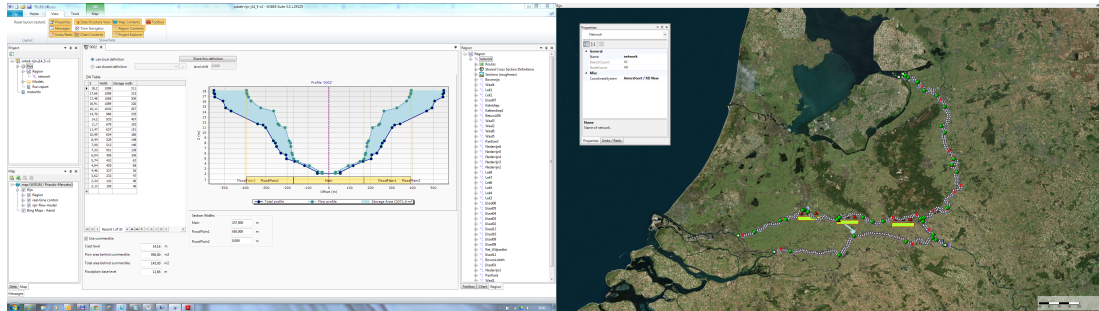
Figure 4.14: Docking windows on two screens within the Delta Shell framework.

4.2.2 Multiple tabs

In case two windows are docked in one view, the underlying window (tab) can be brought to the front by simply selecting the tab, as is shown here.
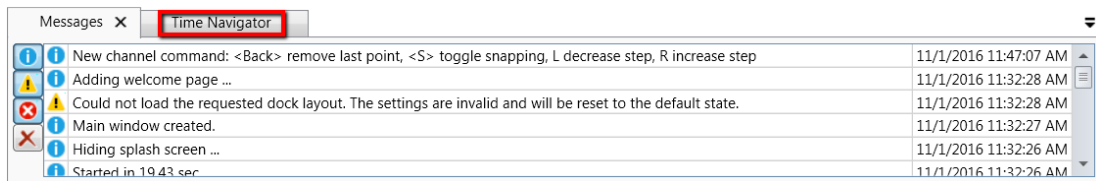
*Figure 4.15: Bringing the **Time Navigator** window to the front*

By dragging dockable windows with the left mouse button and dropping the window left, right, above or below another one the graphical user interface can be customized according to personal preferences. Here an example of the **Time Navigator** window being docked above the **Properties** window.
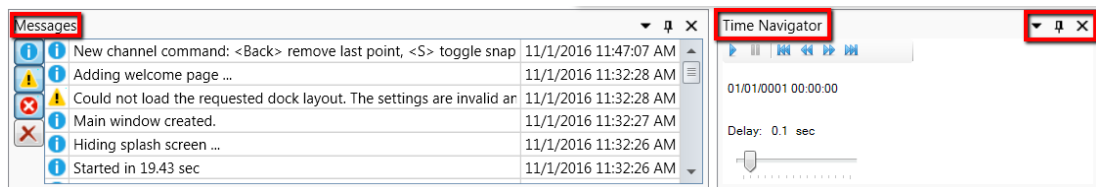
*Figure 4.16: Docking the **Time Navigator** window.*

Additional features are the possibility to remove or (auto) hide the window (top right in Figure 4.16). In case of removal, the window can be retrieved by a mouse-click on *Time Navigator* in the *View* ribbon. Hiding the **Time Navigator** window results in:
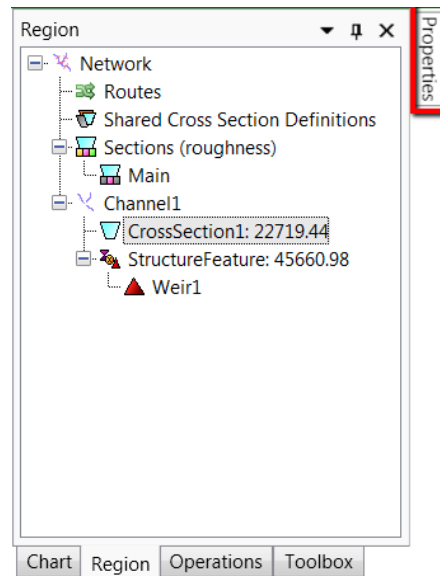
Figure 4.17: *Auto hide the **Properties** window*

4.3 Ribbons and toolbars

The user can access the toolbars arranged in *ribbons*. Model plug-ins can have their own model specific *ribbon*. The *ribbon* may be auto collapsed by activating the *Collapse the Ribbon* button when right-mouse-clicking on the *ribbon*.

4.3.1 Ribbons (shortcut keys)

Delta Shell makes use of ribbons, just like Microsoft Office. You can use these ribbons for most of the operations. With the ribbons comes shortcut key functionality, providing shortcuts to perform operations. If you press `Alt`, you will see the letters and numbers to access the ribbons and the ribbon contents (i.e. operations). For example, `Alt + H` will lead you to the *Home*-ribbon (Figure 4.18).

Note: Implementation of the shortcut key functionality is still work in progress.
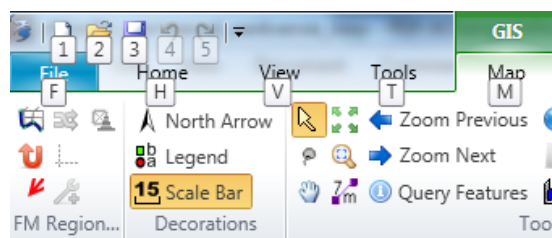
Figure 4.18: *Perform operations using the shortcut keys*

4.3.2 File

The left-most *ribbon* is the *File* ribbon. It has menu-items comparable to most Microsoft applications. Furthermore, it offers users import and export functionality, as well as the *Help* and *Options* dialogs, as shown in Figure 4.19 and Figure 4.20.
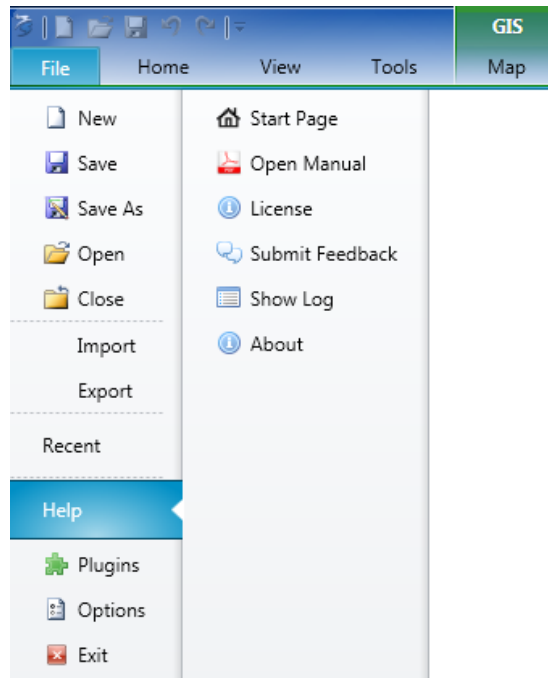
Figure 4.19: *The* File *ribbon.*
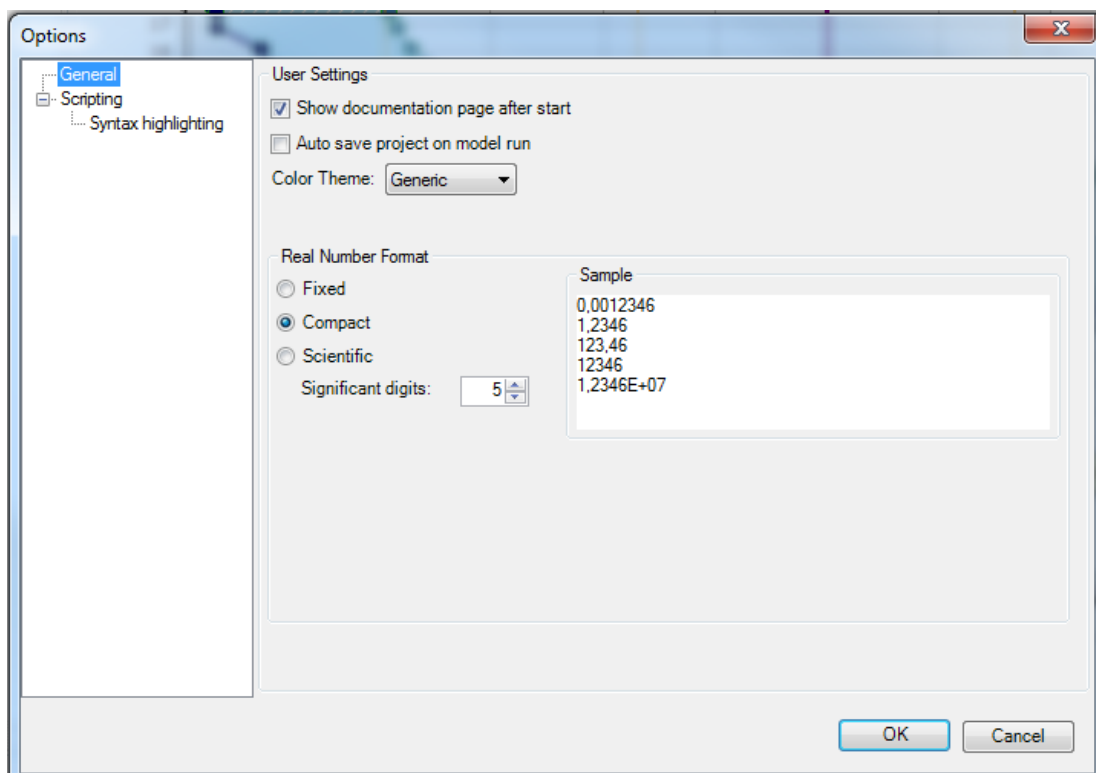
Figure 4.20: *The Delta Shell options dialog.*

4.3.3 Home

The second *ribbon* is the *Home* ribbon (Figure 4.21). It harbours some general features for clipboard actions, addition of items, running models, finding items within projects or views, and help functionality.
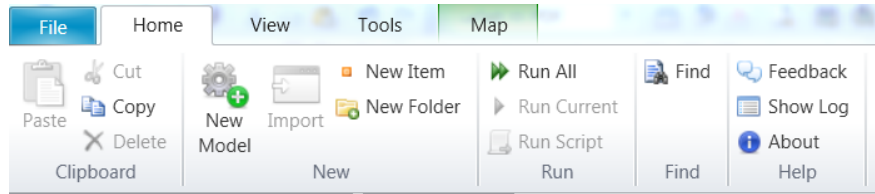
4.3.4 View

The third *ribbon* is the *View* ribbon (Figure 4.22). Here, the user can show or hide windows.
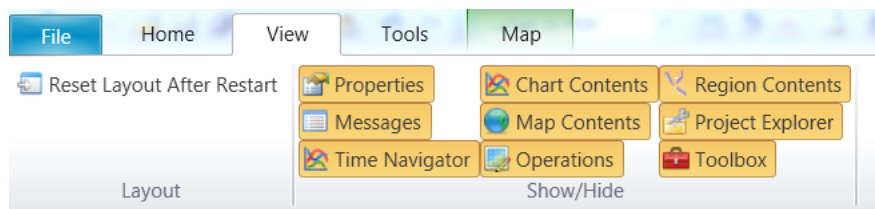
Figure 4.22: The View *ribbon.*

4.3.5 Tools

The fourth *ribbon* is the *Tools* ribbon (Figure 4.23). By default, it contains only the *Open Case Analysis View* tool. Some model plug-ins offer the installation of extra tools that may be installed. These are documented within the user documentation of those model plug-ins.
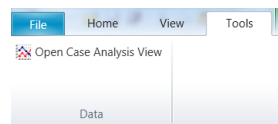
Figure 4.23: The Tools *ribbon contains just the* Data *item.*

4.3.6 Map

The last *ribbon* is the *Map* ribbon (Figure 4.24).
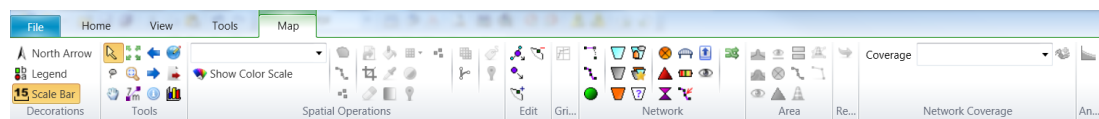
Figure 4.24: The Map *ribbon.*

This will be used heavily, while it harbours all *Geospatial* functions, like:

◇ *Decorations* for the map

         □ North arrow

         □ Scale bar

         □ Legend

         □ ...

    ◇ *Tools* to customize the map view

         □ Select a single item

         □ Select multiple items by drawing a curve

         □ Pan

         □ Zoom to Extents

         □ Zoom by drawing a rectangle

         □ Zoom to Measure distance

         □ ...

    ◇ *Edit* polygons, for example within a network, basin, or waterbody

         □ Move geometry point(s)

         □ Add geometry point(s)

         □ Remove geometry point(s)

    ◇ Creation of a model *Network*, for example for D-Flow 1D

         □ Add new Branch

         □ Split Branch

         □ Add Cross section

         □ Add Weir

         □ Add Pump

         □ ...

Still, all functions of the category can be activated as they will appear in the drop-down panel.

4.3.7 Scripting

When you open the scripting editor in Delta Shell, a Scripting *ribbon* category will appear. This ribbon has the following additional options (see also Figure 4.25), which are described in Table 4.1:
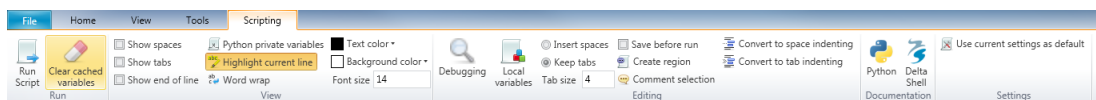
Figure 4.25: *The scripting* ribbon *within Delta Shell.*

Table 4.1: *Functions and their descriptions within the scripting* ribbon *of Delta Shell.*

Function	Description
Run script	Executes the selected text. If no text is selected then it will execute the entire script
Clear cached variables	Clears all variables and loaded libraries from memory

Table 4.1: *Functions and their descriptions within the scripting* ribbon *of Delta Shell.*

Function	Description
Debugging	Enables/Disables the debug option. When enabled you can add breakpoint to the code (using `F9` or clicking in the margin) and the code will stop at this point before executing the statement (use F10 (step over) or F11 (step into) for a more step by step approach)
Python variables	Show or hide python variables (like _var_) in code completion
Insert spaces/tabs	Determines if spaces or tab characters are added when pressing tab
Tab size	Sets the number of spaces that are considered equal to a tab character
Save before run	Saves the changes to the file before running
Create region	Creates a new region surrounding the selected text
Comment selection	Comments out the selected text
Convert to space indenting	Converts all tab characters in the script to spaces. The number of spaces is determined by Tab size
Convert to tab indenting	Converts all x number of space characters (determined by Tab size) in the script to tabs
Python (documentation)	Opens a link to the python website, showing you the python syntax and standard libraries
Delta Shell (documentation)	Opens a link to the Delta Shell documentation website (generated documentation of the Delta Shell api)

4.3.8 Shortcuts

The shortcut keys of the scripting editor within Delta Shell are documented in Table 4.2.

Table 4.2: *Shortcut keys within the scripting editor of Delta Shell.*

Shortcut	Function
`Ctrl + Enter`	Run selection (or entire script with no selection)
`Ctrl + Shift + Enter`	Run current region (region where the cursor is in)
`Ctrl + X`	Cut selection
`Ctrl + C`	Copy selection
`Ctrl + V`	Paste selection
`Ctrl + S`	Save script
`Ctrl + -`	Collapse all regions
`Ctrl + +`	Expand all regions
`Ctrl + "`	Comment or Uncomment current selection
`Ctrl + W`	Add selection as watch variable
`Ctrl + H`	Highlight current selection in script (press esc to cancel)
`F9`	Add/remove breakpoint (In debug mode only)
`F5`	Continue running (In debug mode only — when on breakpoint)
`Shift + F5`	Stop running (In debug mode only — when on breakpoint)
`F10`	Step over current line and break on next line (In debug mode only - when on breakpoint)

Table 4.2: Shortcut keys within the scripting editor of Delta Shell.

Shortcut	Function
F11	Step into current line if possible, otherwise go to next line (In debug mode only — when on breakpoint). This is used to debug functions declared in the same script (that have already runned)

4.3.9 Quick access toolbar

Note: The user can make frequently used functions available by a single mouse-click in the *Quick Access Toolbar*, the top-most part of the application-window. Do this by right-mouse-clicking a ribbon item and selecting *Add to Quick Access Toolbar*.
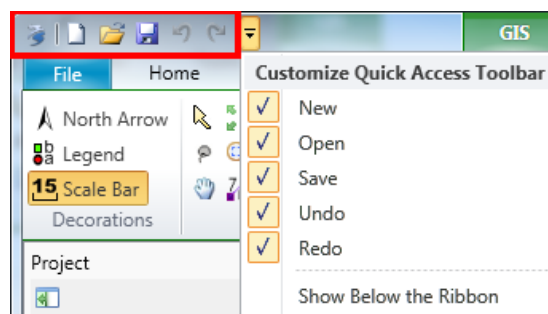
Figure 4.26: The quick access toolbar.

5 Module D-Geo Flow: Getting started

5.1 Starting a D-Geo Flow model

In this chapter it is explained how to get started with D-Geo Flow. The tutorial describes general procedures for the creation of a geometry model, the assignment of soil properties and boundary conditions, the generation of a finite element mesh, the execution of a calculation and the visualisation of the results.

5.1.1 Problem definition

In this tutorial a first application is considered, namely the piping mechanism under a dike.
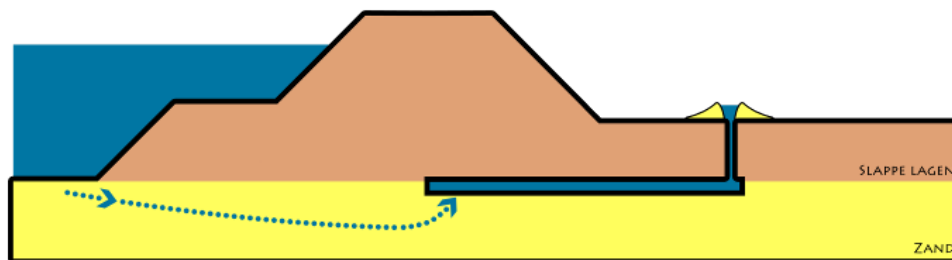
Figure 5.1: *Pipe Formation.*

A dike is built on a sandy aquifer covered by a cohesive top layer. The water level in the river varies with time. The hydraulic gradient across the structure results in a groundwater flow through the aquifer. Transient flow calculations with D-Geo Flow define the formation of a pipe at the polder side and its backwards development from the toe up to the river side.

5.1.2 Start a new project

Open the Delta Shell GUI. On the Home tab, click on New Model

Choose a D-Geo Flow Model from the available models
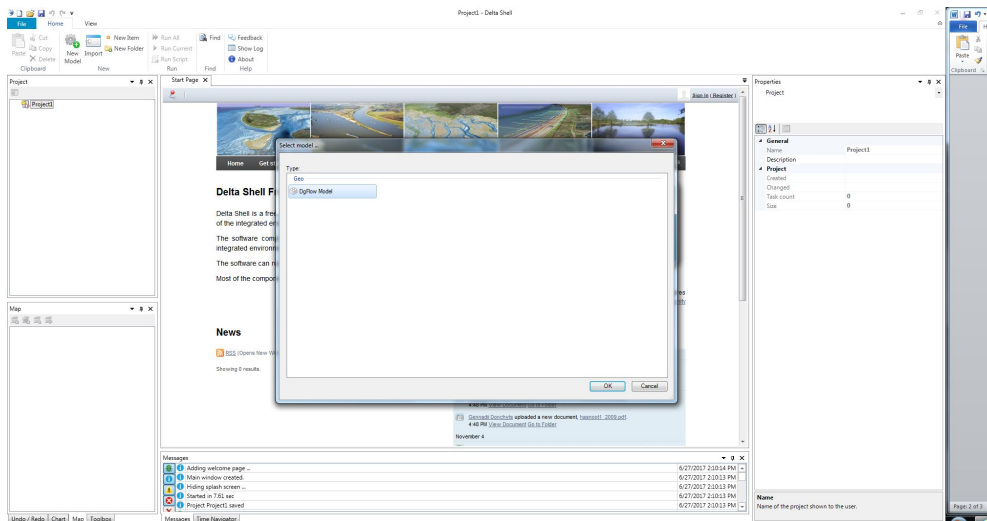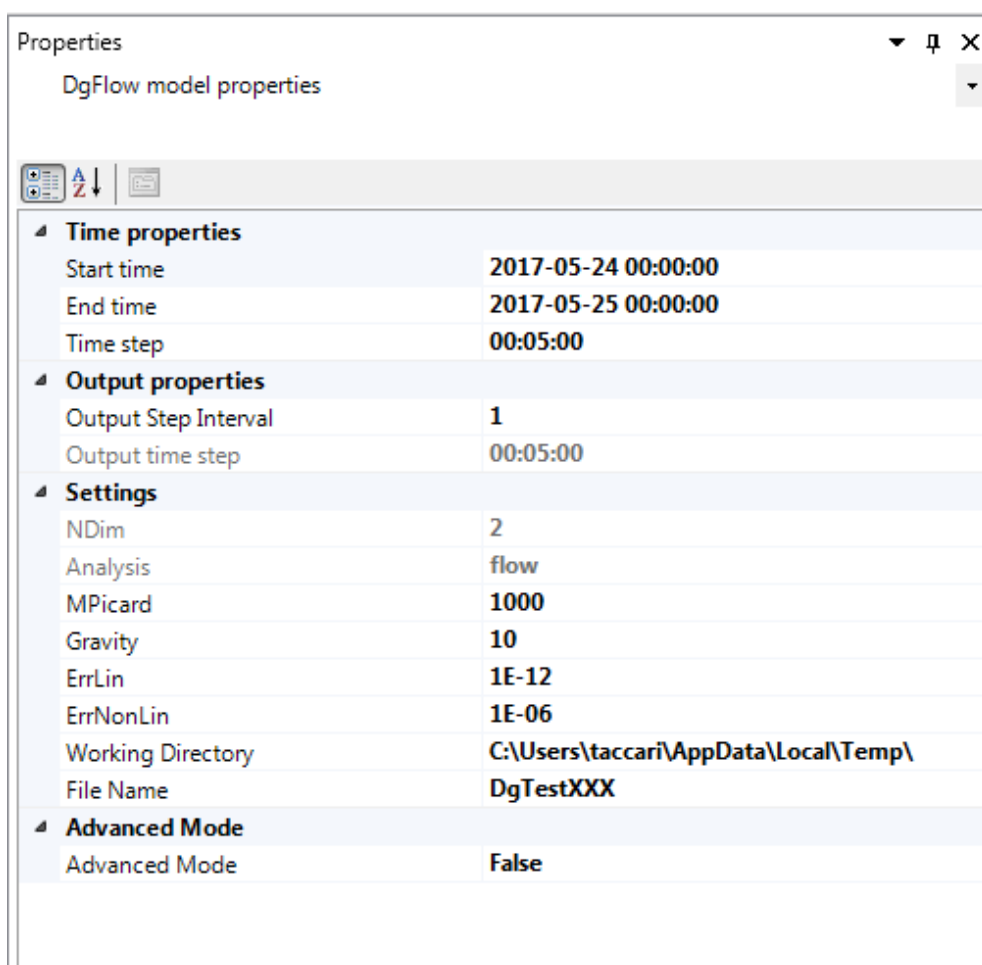
Figure 5.2: *Pipe Formation.*

◇ Under the *View* ribbon, at the top of the window, you can select the boxes that you want to visualize. Ensure that all the boxes are selected.

◇ In the *project* box, extend the tab *Project1 → D-Geo Flow Model*. Click on *D-Geo Flow Model* icon.

◇ You can rename your model by a right mouse click on D-Geo Flow Model (the standard name for a new model) and choosing Rename. Let's call our model "Piping"

5.2 Geometry schematization

5.2.1 Model properties

◇ Click on *D-Geo Flow Model* icon.

◇ Insert the model properties in the *Properties* box, as indicated in the figure below. The properties can be visualized either in categorized or alphabetical order.

Figure 5.3: Model properties.

The total calculation time is 1 day and the output results are produced every 5 minutes. The results are visualized at every interval, meaning that the output time step results equal to the time step used in the calculation. The meaning of MPicard, ErrLin and ErrNonLin is further explained in 'Scientific background' on page 55. The working directory indicates where the results are temporary stored. The results will be stored in a folder with the same name as the file name. The advance mode is set to False.

5.2.2 Soil stratigraphy

5.2.2.1 Soil parameters

◇ In the *project* box, extend the tab *Project1* → *Piping* → *Input* → *Materials*. Right-click on the *soil* folder and select *Add new soil*.
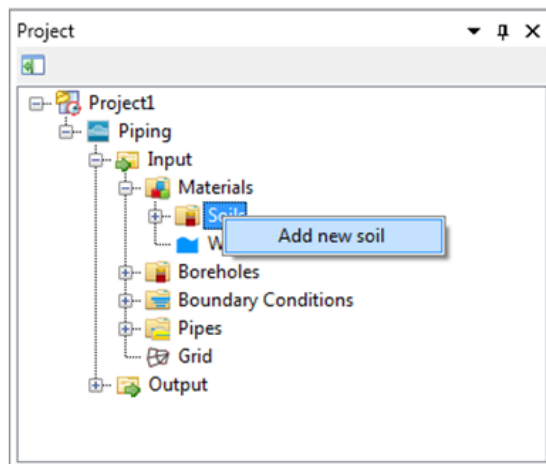
Figure 5.4: *Add new soil.*
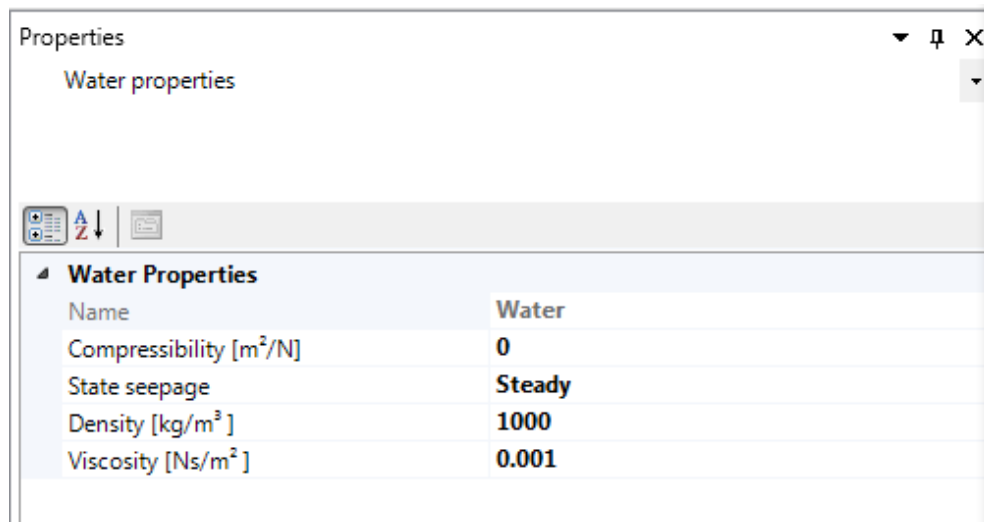
◇ In the same way, create four more soil materials.

◇ Extend the *Soil* tab and click on *New Soil* that you just created.

◇ Insert the following properties in the *Properties* box:

Table 5.1: *Material properties of the New Soils.*

General					
Name	Blanket	Dike	Aquifer top	Aquifer bottom	Blanket polder side
Color	244; 164; 96	250; 128; 114	255; 228; 225	160; 82; 45	255; 215; 0
Other					
Porosity [-]	0.5	0.4	0.3	0.5	0.5
Compressibility [m^2/N]	0	0	0	0	0
Submerged Particle Density [kg/m^3]	0	0	1650	0	0
D70 [m]	0	0	0.0002	0	0
η [-]	0	0	0.25	0	0
θ [deg]	0	0	37	0	0
Permeability					
Hydraulic conductivity - X [m/day]	0.01	0.01	10	30	0.01
Hydraulic conductivity - Y [m/day]	0.01	0.01	10	30	0.01

5.2.2.2 Water properties

◇ In the *project* box, extend the tab *Project1 → Piping → Input → Materials*. Click on the *Water* folder.

◇ Check that the following properties are assigned to the water:

Figure 5.5: *Water properties.*

5.2.2.3 Soil layering (in boreholes)

◇ Right-click on the *Boreholes* folder and select *Add new borehole*. Double click on the borehole just created and a new window will open.

◇ The first borehole is located at the x-coordinate equal to 0. Its top level is at -2 m and it consists of two layers, namely *Aquifer top* from -2 m to -12 m and *Aquifer bottom* from -12 m to -20 m.

◇ Place the pointer on top of the borehole and select *Aquifer Top* to assign this material to the borehole.

◇ Right-click on the borehole in order to add new layers. Set the *Top level* at z = -2 m and the first Level at z = -12 m. Assign *Aquifer bottom* to the second layer and set the deepest level at -20m. Keep the X position at x = 0.
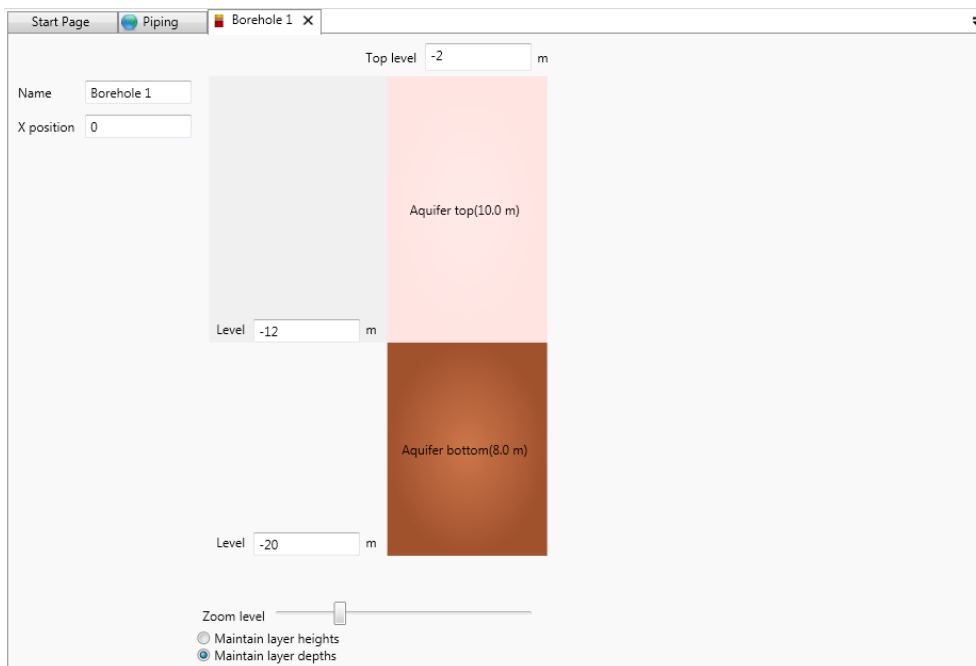
Figure 5.6: *The generated borehole in the Borehole 1 window.*

◇ Add a second borehole. You can do it in two ways: you can right-click on the borehole folder to add a new borehole as previously done, or you can right click on borehole 1 and select *Create copy of Borehole 1*. In this way, a new borehole, identical to Borehole 1, will be created.
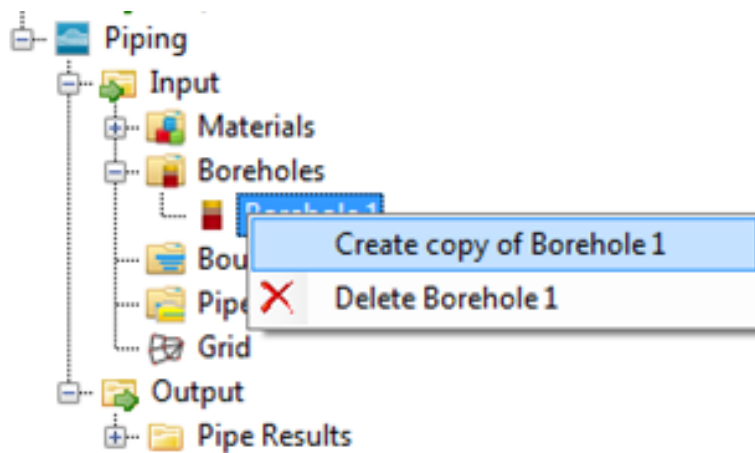
Figure 5.7: *Add a new borehole, equal to the borehole just created.*

◇ In this way, create 12 boreholes as given in the following table:

Table 5.2: *Boreholes of the model.*

Borehole number	X position	Top Level	Material 1st Layer	Level	Material 2nd Layer	Level	Material 3rd Layer	Level	Material 4th Layer	Level
1	0	-2	Aquifer top	-12	Aquifer bottom	-20				
2	6	-2	Aquifer top	-12	Aquifer bottom	-20				
3	10	0	Blanket	-2	Aquifer top	-12	Aquifer bottom	-20		
4	20	0	Blanket	-2	Aquifer top	-12	Aquifer bottom	-20		
5	32.5	5	Dike	0	Blanket	-2	Aquifer top	-12	Aquifer bottom	-20
6	37.5	5	Dike	0	Blanket	-2	Aquifer top	-12	Aquifer bottom	-20
7	48	0	Blanket	-2	Aquifer top	-12	Aquifer bottom	-20		
8	58.25	0	Blanket	-2	Aquifer top	-12	Aquifer bottom	-20		
9	60	-2	Aquifer top	-12	Aquifer bottom	-20				
10	63.75	-2	Aquifer top	-12	Aquifer bottom	-20				
11	67.5	0	Blanket polder side	-2	Aquifer top	-12	Aquifer bottom	-20		
12	100	0	Blanket polder side	-2	Aquifer top	-12	Aquifer bottom	-20		

⬦ Once you create the 12 boreholes, right click on the *Boreholes* folder and select *Create cross-section*.

Please note that you can create a cross section also by creating a geometry of the model. You can do so by selecting Draw rectangle or dray polygon, under the Map ribbon, at the top of the window.
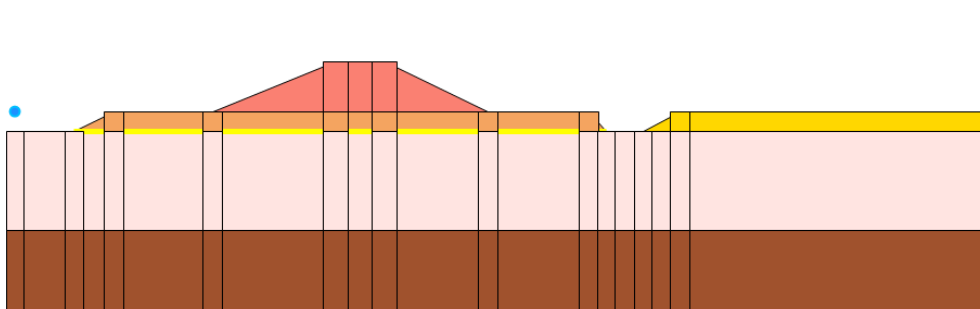
Figure 5.8: *The generated cross section.*

5.2.3 Boundary conditions

5.2.3.1 Types of boundary conditions

⬦ Right-click on the *Boundary Conditions* folder in the *project* box and select *Add submerging boundary condition*.

⬦ Click in *Submerging Boundary1* , that you just created, and, in the *Properties* box, set the name to Riverside head boundary and the color to 75; 110; 187

◇ We assign the boundary condition in the form of head. Return to the *project* box and, under the *Riverside head boundary* tab, double click on *Head*.
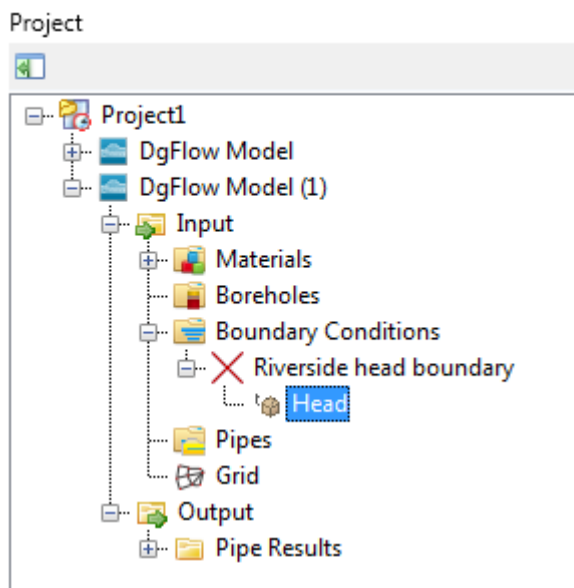
Figure 5.9: Add Riverside head boundary as Head in the boundary conditions of the model.

◇ A new window appears. Select Table and Chart (single function)
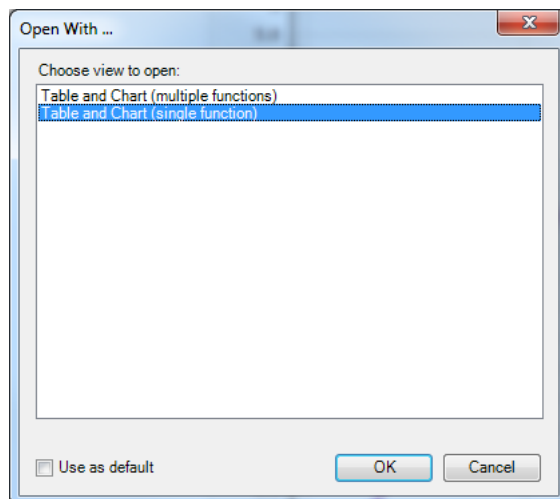
Figure 5.10: Select Table and Chart (single function).

◇ In the *Head* tab insert the boundary condition as given below:

Table 5.3: Boundary condition for the Riverside head boundary in the form of head.

Time [yyyy-MM-dd HH:mm:ss]	Head [m]
24-05-17 00:00	2
24-05-17 12:00	6.5
25-05-17 00:00	2
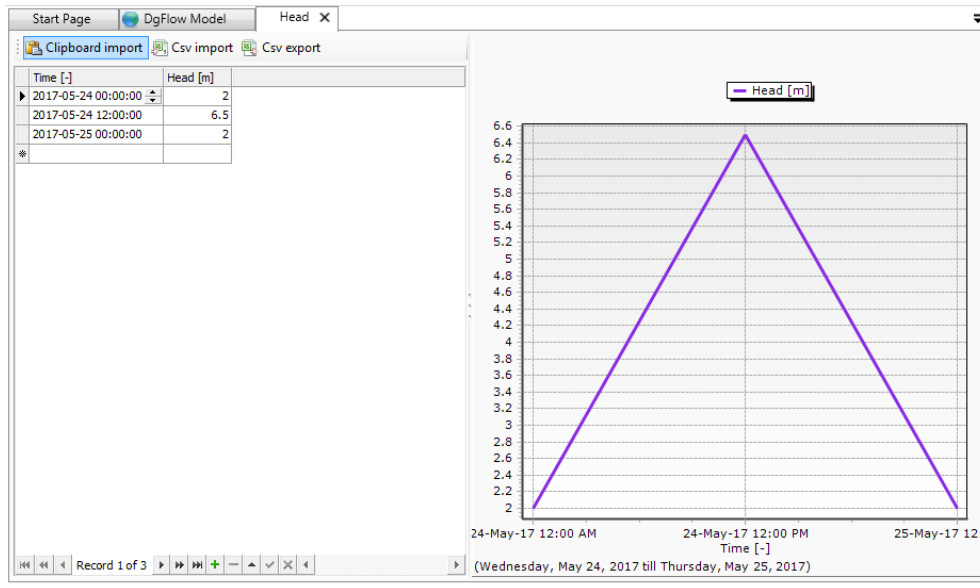
Figure 5.11: Riverside head boundary in the Head window.

◇ Add a second boundary condition: select *Add head boundary condition*. In the *Properties* box, set the name to Polder level and the color to 186; 251; 97

◇ In the same way as described above. Set the following Time and Head:

Table 5.4: Second boundary condition in the form of head.

Time [yyyy-MM-dd HH:mm:ss]	Head [m]
24-05-17 00:00	0
25-05-17 00:00	0

◇ You still need to define two boundary conditions. The first is a seepage boundary condition. Therefore select Add seepage boundary condition. Rename it to 'Blanket polder side'.

◇ The last boundary condition that needs to be assigned is the default no boundary condition to the boundaries where no conditions are applied. This will be done later, first we need to assign the boundary condition to the boundaries of our model.

5.2.3.2 Assigning boundary conditions

Now you have to assign the boundary condition to the model. You need to assign the boundary conditions that you just created at the top of the model, as indicated in the figure below. The default (no flow) boundary conditions will be then assigned to the three remaining sides of the model.
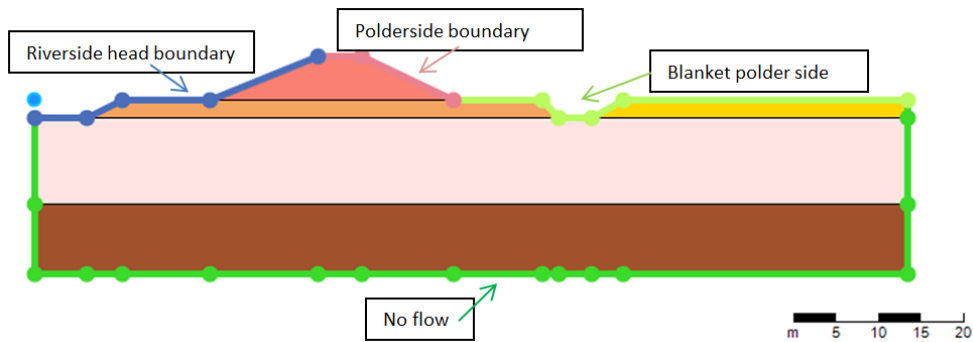
Figure 5.12: *Boundary conditions of the model.*

◇ Go the the box Map → extend the list → uncheck the Boreholes checkbox. Do this in order to make the boreholes invisible so all the boundaries can be selected .
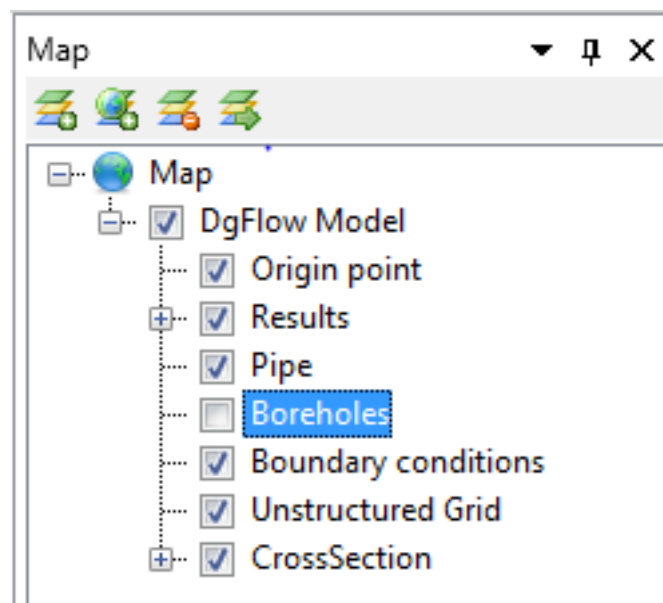
Figure 5.13: *Location of the Boreholes checkbox.*

◇ Select the line at the top left of the model. This line is defined as LINESTRING (0 -2, 6 -2) from the x and y- coordinates of its two points. In the *Properties* box, set the Boundary condistion to Riverside head boundary.
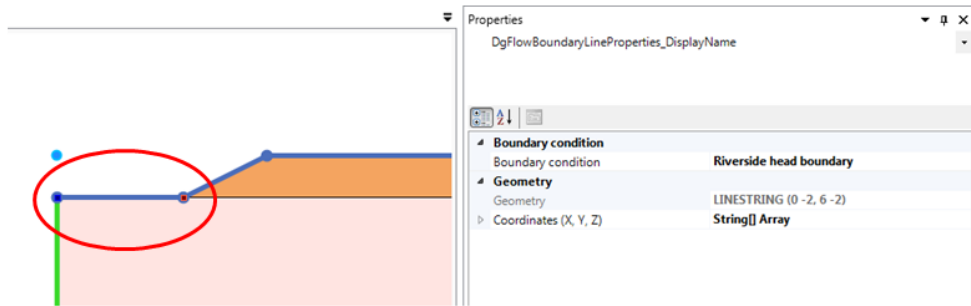
Figure 5.14: Select a line and assign the boundary condition in the Properties box.

◇ Alternatively, you can left-click on the mouse and select an area. In such a way you can assign a boundary condition to multiple lines.
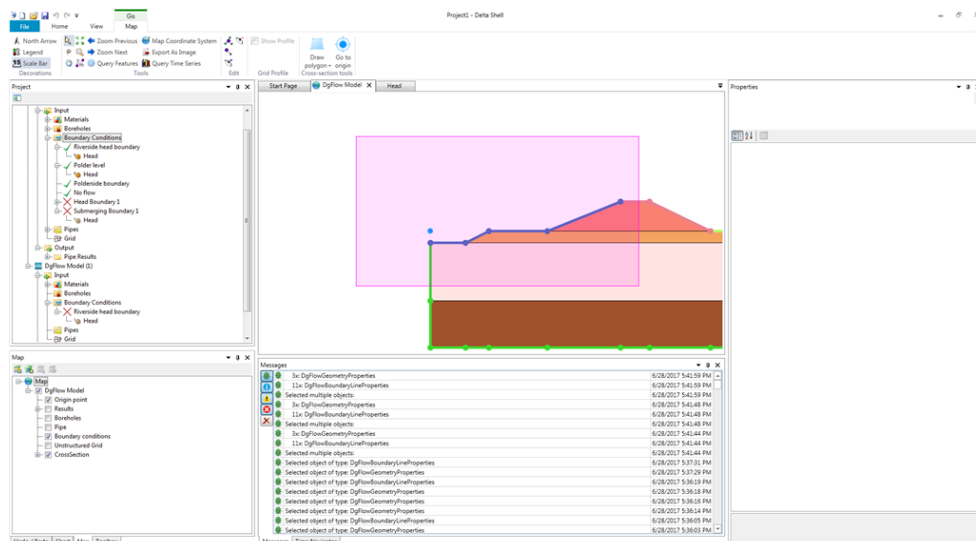
Figure 5.15: Assign the boundary condition by selecting an area.

Pay attention that the boundary conditions have to be assigned both to the lines and to the points. If boundary conditions are not assigned to some points, select the point and assign the boundary condition.

◇ Once boundary conditions are assigned to the top of the model, you can assign default no flow boundary condition to the remaining boundaries. In order to do so, right-click on the *Boundary Conditions* folder in the *project* box and select *Add default (no flow) boundary conditions*. Rename this boundary condition as "No Flow" and assign the color (59; 218; 43)
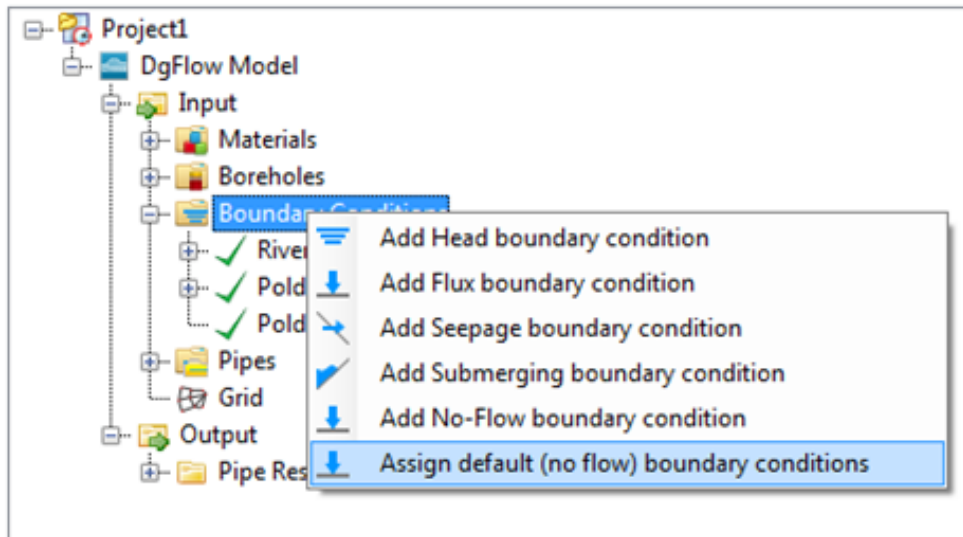
Figure 5.16: *Assign default (no flow) boundary conditions.*

Now all the boundaries of the model have assigned boundary conditions.

5.2.4 Pipe generation

◇ Right-click on the *Pipe* folder in the *project* box and select *Add new pipe*.

◇ Click in *Pipe* , that you just created, and, in the *Properties* box, set the name to start X position to 6, the exit X position to 60 and the Y-Position to -2
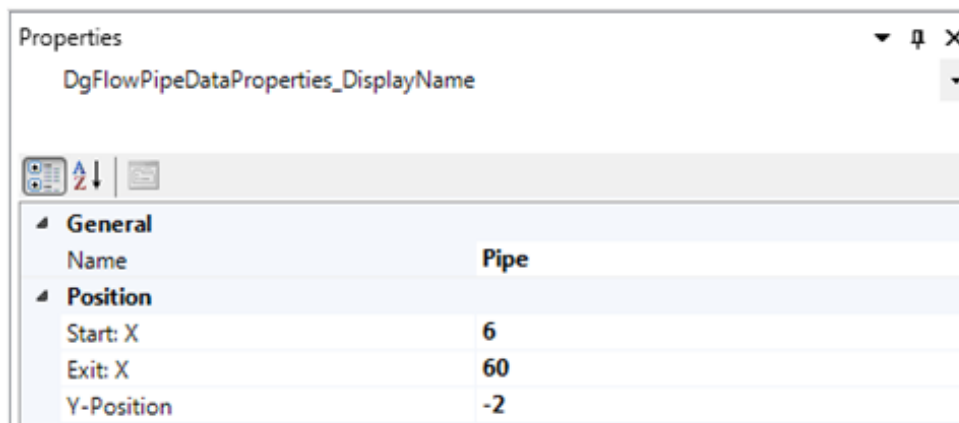
Figure 5.17: *Properties of the pipe.*

The pipe runs horizontally from the dike toe at river to the bottom of the ditch to the dike side.
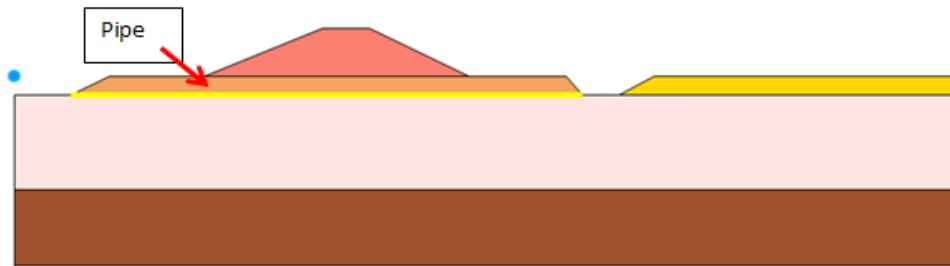
Figure 5.18: Location of the pipe in the model.

5.2.5 Mesh generation

◇ Click on the *Grid* folder in the *project* box.

◇ In the *project* box, assign the coarseness of the mesh equal to 2.5 and the Pipe Coarseness equal to 6.
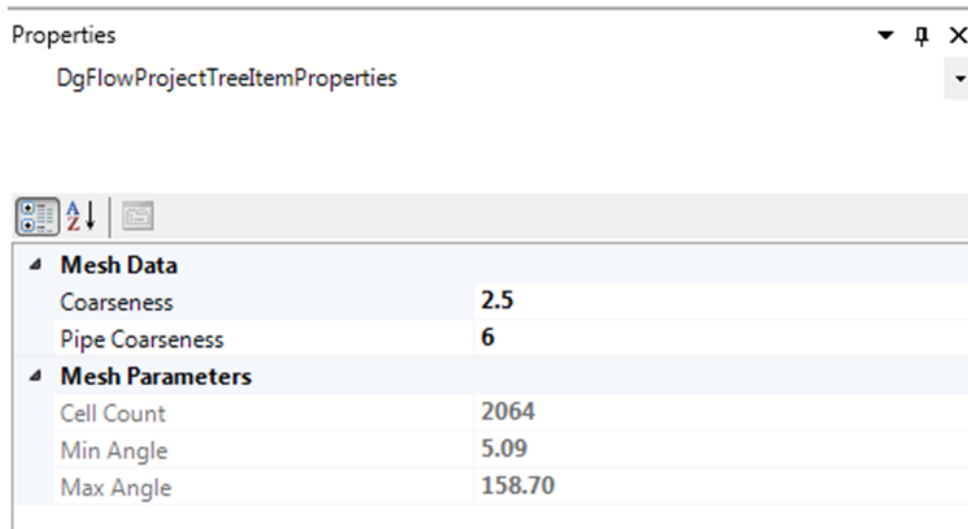
Figure 5.19: Mesh properties.

◇ Now you can generate the mesh. Right-click on the *Grid* folder in the *project* box and select *Generate grid*.
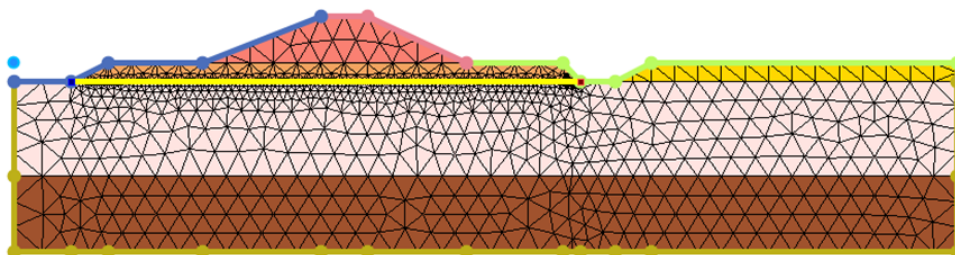
Figure 5.20: Mesh in the model

5.3 Performing calculations

Before performing the calculation, you can validate the model in order to check if the model is ready and calculations can be carried on.

◇ In the *project* box, right-click on the project tab and select *Validate*. A new window appears: all the inputs have a green mark. The model is validated and calculations can be done.
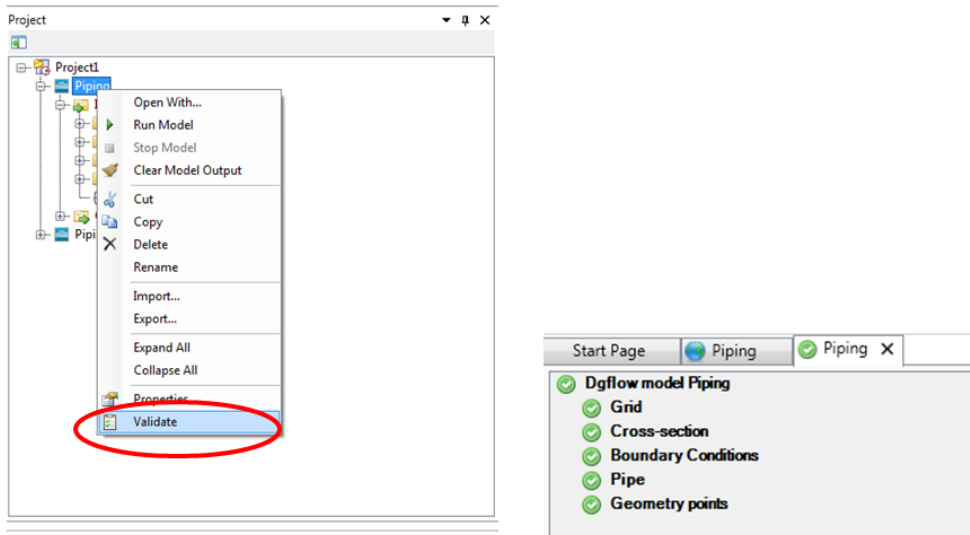
Figure 5.21: *Validation of the model.*

◇ In the *project* box, right-click on the project tab and select *Run model*.
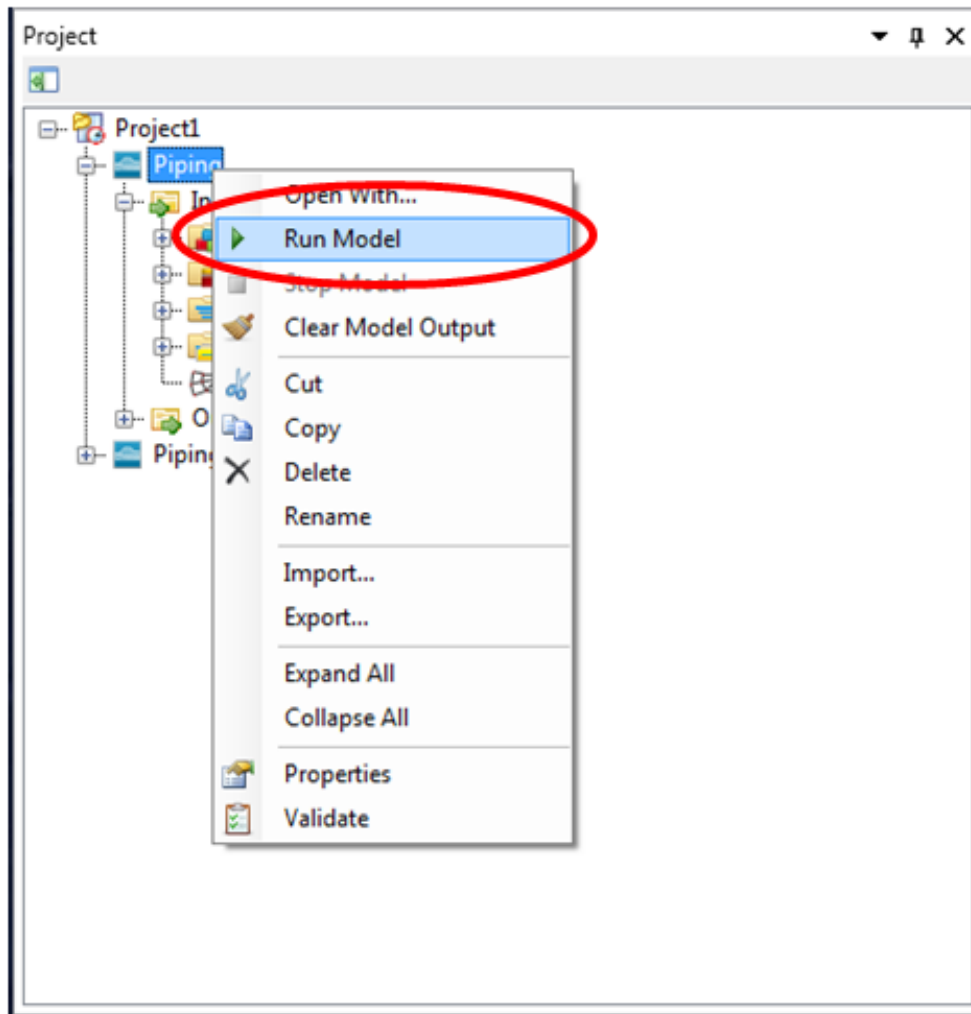
Figure 5.22: Run the model.

◇ A new window appears, which indicates the progress of the calculation.
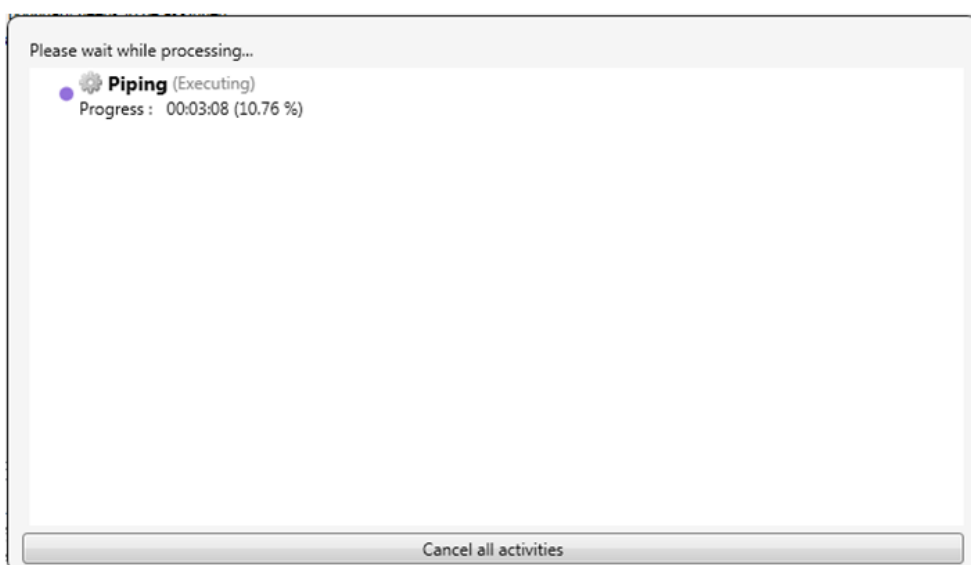
Figure 5.23: The model is running.

5.4 Visualization of results

There are several ways in which the results of the calculations can be visualised. This chapter describes how to do so.

5.4.1 Point and grid results

The first way to visualize results is to select a point and obtain the results for that specific point. In the *Map* box, unselect the *Results*.
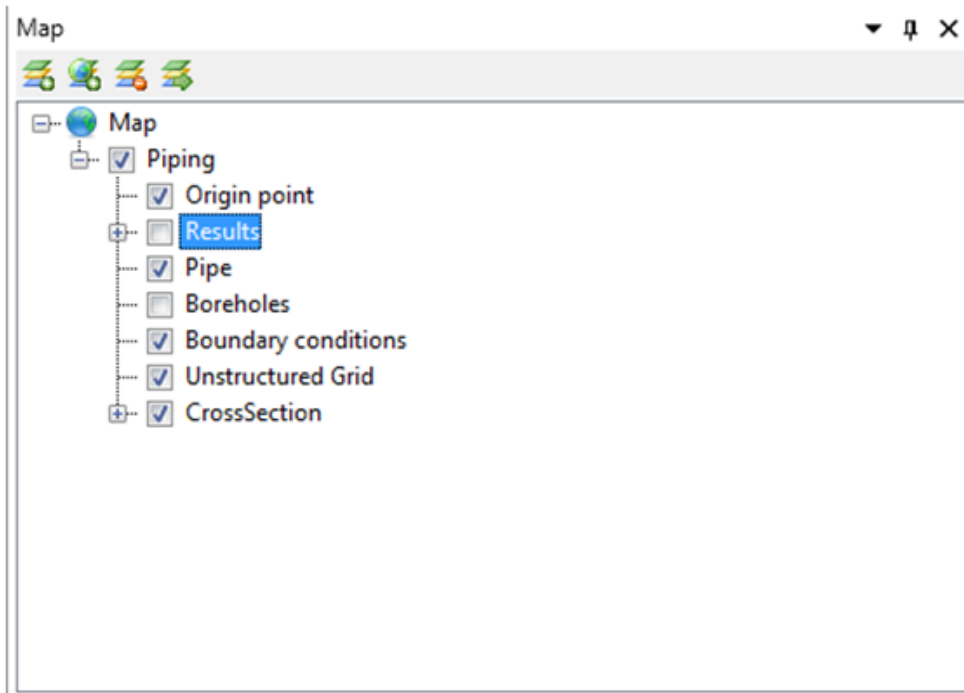
Figure 5.24: *Unselect the results in the Map box.*

Now you can see the grid and select a point. Right-click and select *Query time series*.
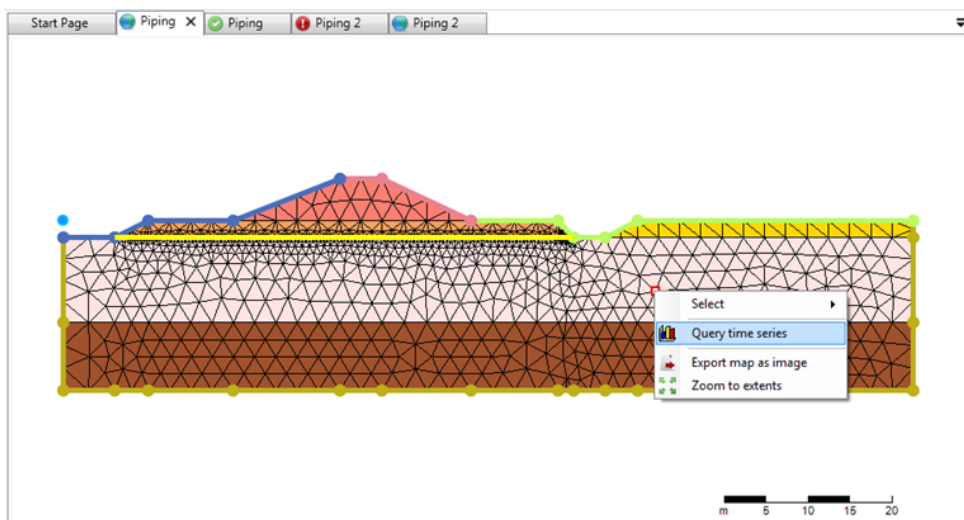
Figure 5.25: *Selection of a point for visualization of results.*

Select the results that you want to visualize, such as for example *Head*.
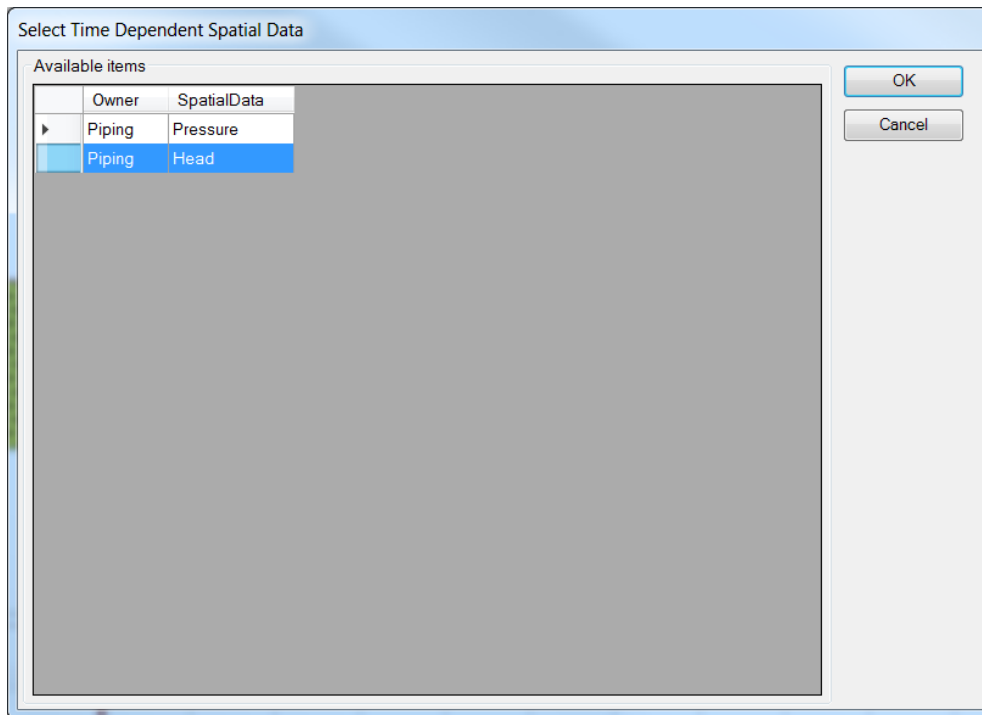
Figure 5.26: *Selection of results to visualize.*

You will now obtain a graph with Time in the x-axis and Head in the vertical axes.

At the same way, you can visualize the results for an element of the grid. Right-click on an element and select *Query time series*, as previously done. You can visualize the results in term of permeability in x direction, permeability in y direction and velocity
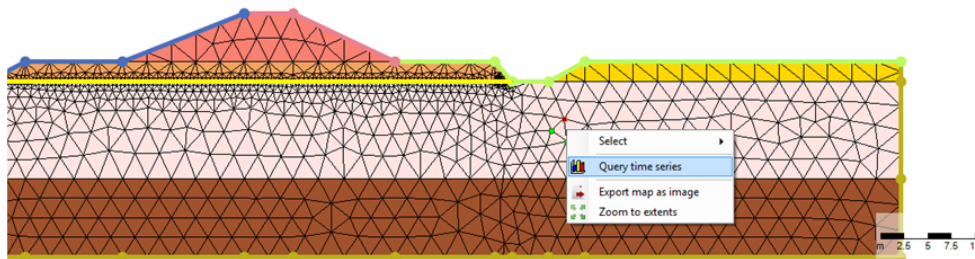
Figure 5.27: *Selection of an element of the grid for visualization of results.*

5.4.2 Profile results

You can also visualize the results along a profile for a specific output, while using the Time Navigator.

◇ In the Map box, select only the Result that you want to visualize. Select, for example, Head.

◇ In the Map tab sheet , at the top of the window, select Show Profile

◇ Left-click at the left of the profile near point (0;-10) and double left-click at the right of the profile, around point (100; -2).
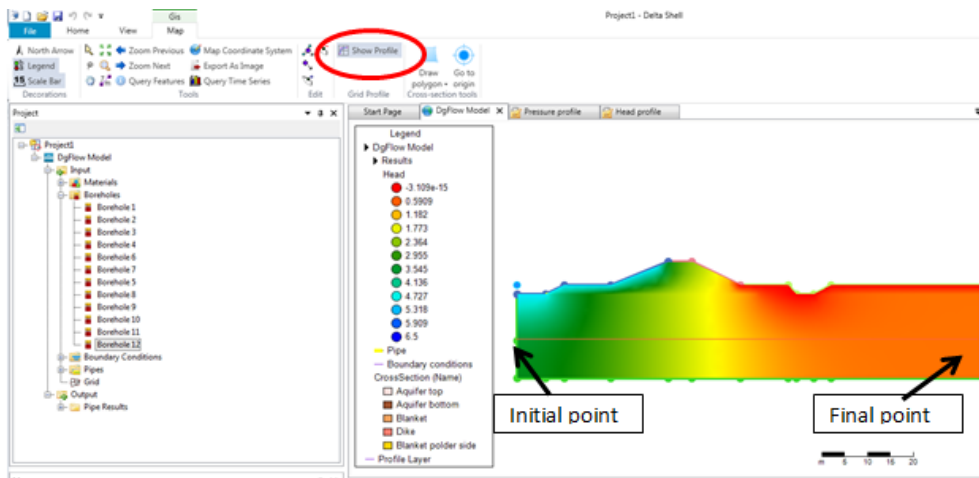
Figure 5.28: *Definition of Profile*

◇ A new window Head profile will open with the values of the head along the chosen profile.
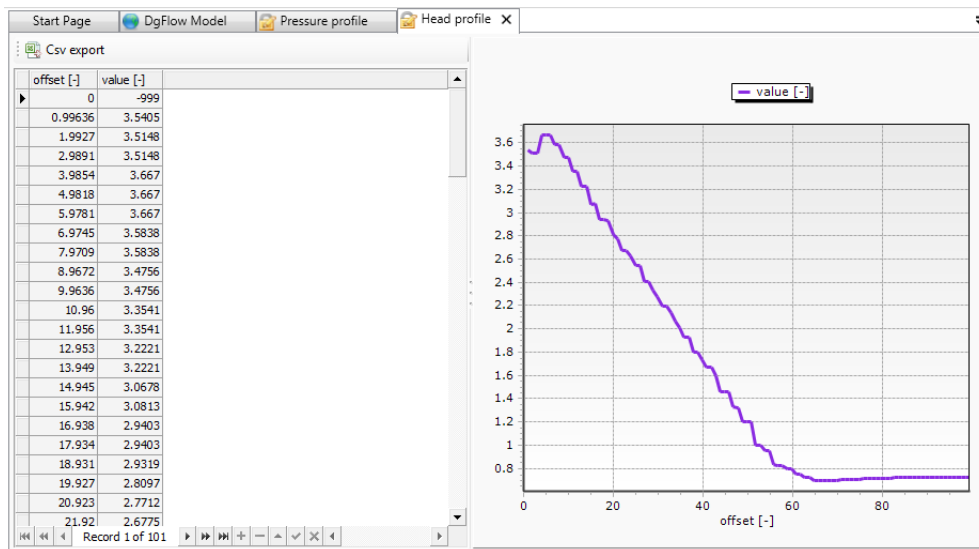
Figure 5.29: *Definition of Profile*

5.4.3 Time Navigator

Another way to visualize the results is to see how the parameters are changing in time within the whole model.

In the *Map* box, select the Results in order to make them visible. Extend the tab *Results* and select only *Head*.
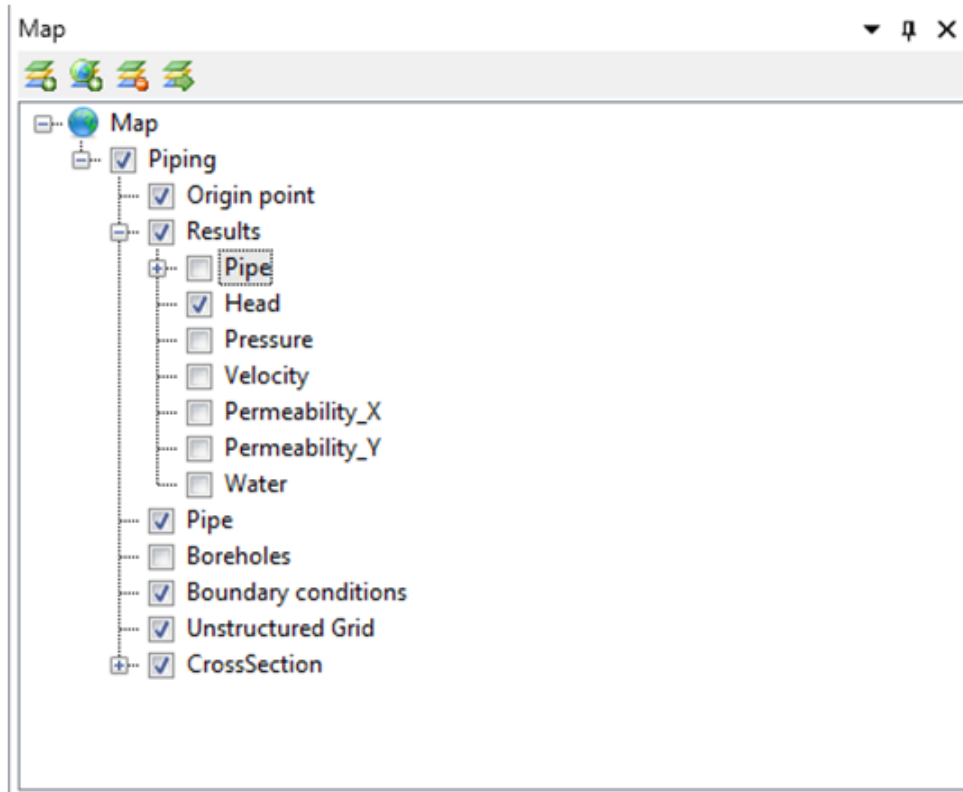
Figure 5.30: *Selection of Head in the Results.*

In the *Time Navigator* window, you can run a simulation which shows the results in the main window.

Figure 5.31: *Time Navigator.*

5.4.4 Piping results

The results for the pipe are available in a different window.

◇ In the *project* box, extend the tab *Project1* → *Piping* → *Output* → *Pipe results*. Here you find the results for the pipe.
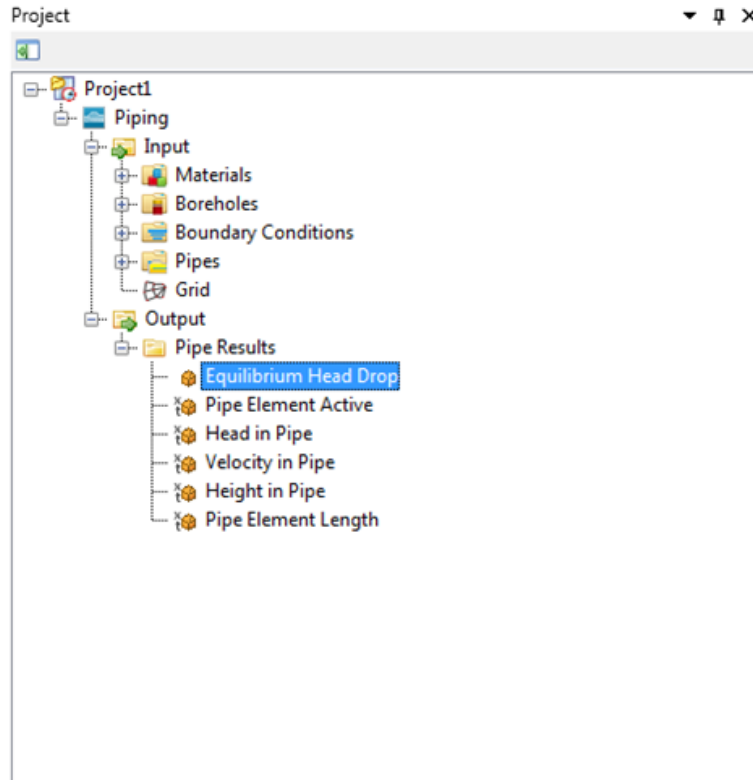
Figure 5.32: *Pipe results.*

Double click on *Equilibrium Head Drop*. A new window appears asking if you want to visualize the results in a table and chart with multiple functions or with only one function. Select the second option, so that every result will be plotted in a different chart.

You will now see a graph with the applied Head Drop [m] versus the Pipe Length [m].
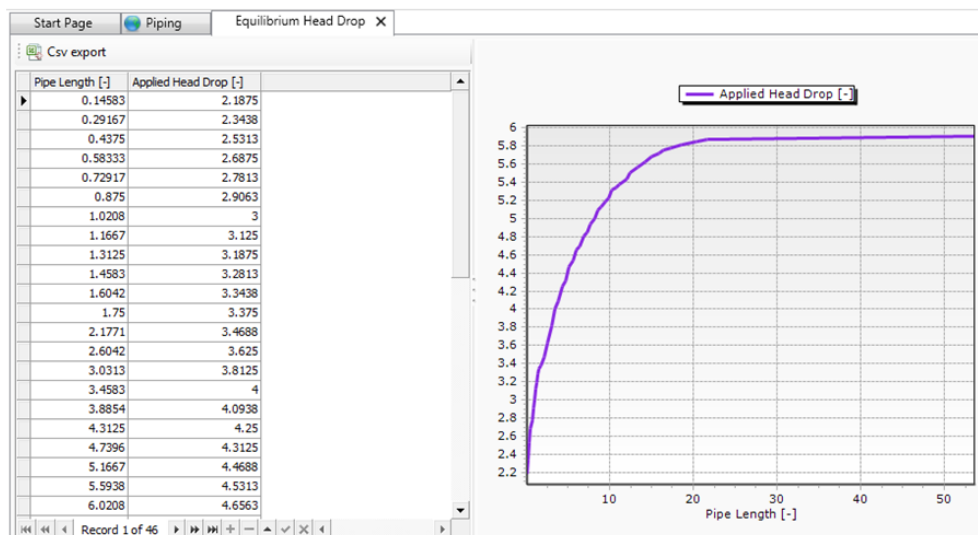
Figure 5.33: *Results for the Equilibrium Head Drop.*

5.4.5 Export the results

There are two ways in which you can export the results:

1 Export the results in a Csv file by selecting the *Csv export tab* at the left top of the result window.

2 Right click on the chart and select *Export as Image..*.
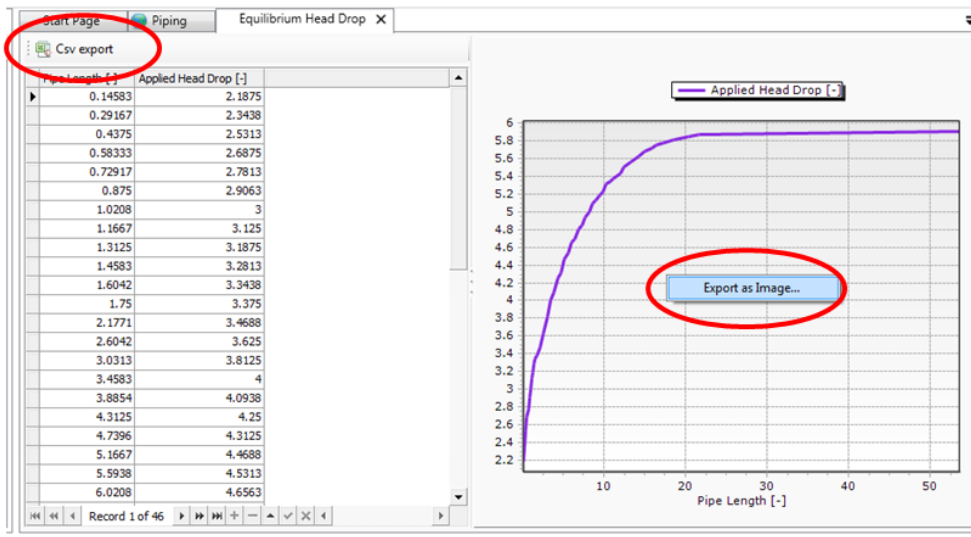
Figure 5.34: *Export the results.*

5.5 Saving a D-Geo Flow model

You can save the model by selecting the save icon at the top of the Delta Shell GUI.
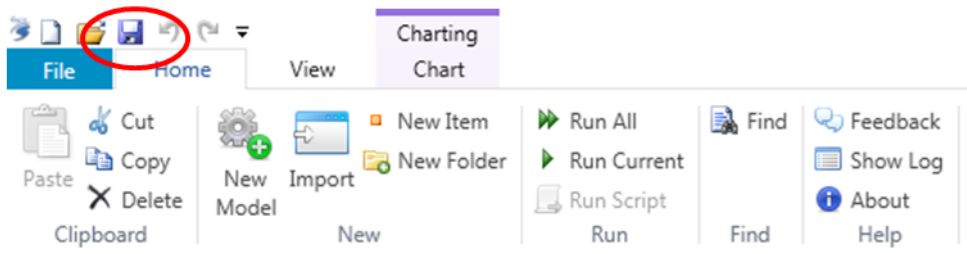
Figure 5.35: *Save a model.*

5.6 Opening an existing D-Geo Flow model

If you, instead, want to open an existing model, you can do so in two ways. You can either select the *Import* tab in the Home ribbon or you can right-click in the tab *Project1* in the *project* box and select *Import*.
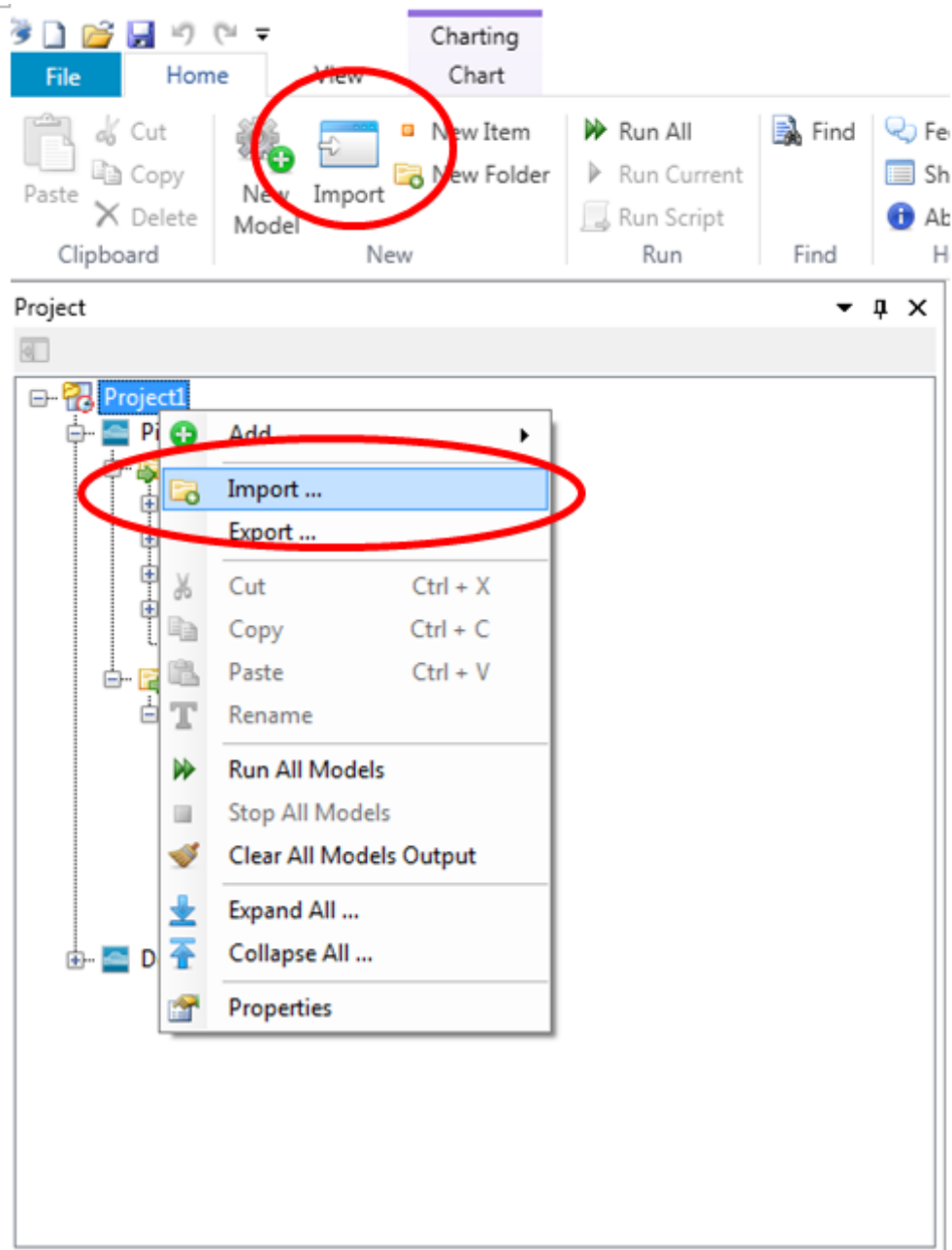
Figure 5.36: *Import a model.*

Choose *D-Geo Flow* model importer from the available data.
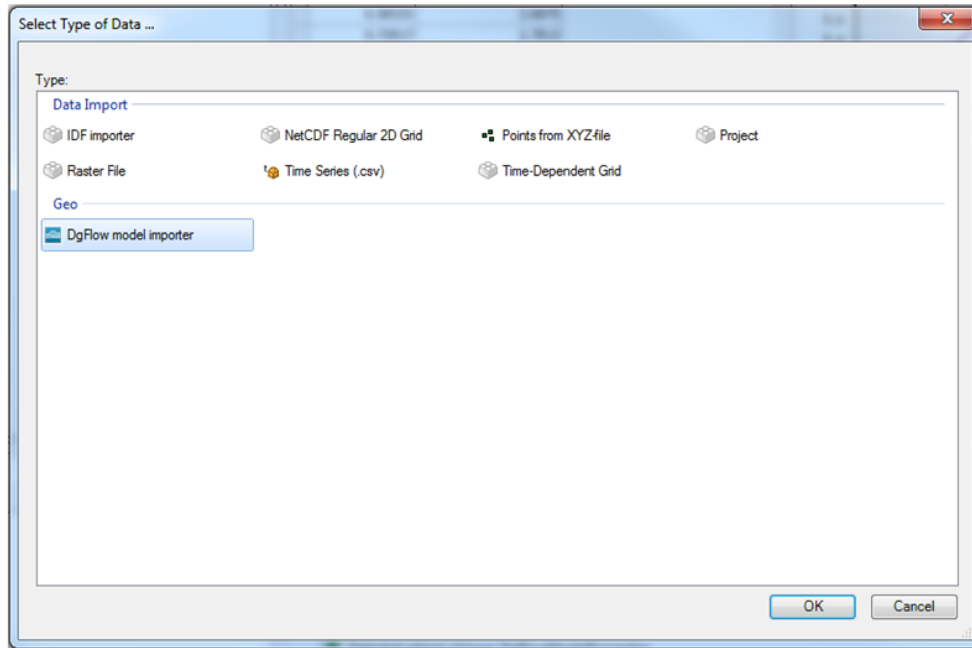
Figure 5.37: *Open D-Geo Flow model importer.*

You can now select the file that you want to open.

6 D-Geo Flow Tutorial Cases

6.1 Description of the cases

This tutorial provides two examples for D-Geo Flow: a river dike and a sea dike. General remarks are:

◇ In the current document the decimal point is used as a decimal separator (conform international notations), note that D-Geo Flow requires a decimal point in your 'regional settings' (Windows).

◇ Note that when 2 or more D-Geo Flow models are open in the same project, the save function will only save the changes made to the project that is highlighted in the project window.

◇ D-Geo Flow offers transient flow piping analysis. The results of these kind of analyses are currently still subject of investigation. It is therefore recommended to start with incompressible flow analysis and consult an expert before conducting transient flow piping calculations.

6.2 Pilot Case 1: River dike

6.2.1 Start-up project

1. Start-up Dg Flow
2. In the top taskbar, click on File → New
3. In the Home tab, in the Project window, right-click on the project and click on add → New Model . . .
4. Select the D-Geo Flow Model and press on OK.
5. On the top taskbar click on File → Save As and safe your project as River_dike_tutorial in your preferred folder.

6.2.2 Input

1. In the Project window, go to River_dike_tutorial → D-Geo Flow Model. In the properties window, set the Start time to: 2017/03/10 00:00:00; set the End time to: 2017/03/11 00:00:00; set the Time step to: 0d 00:05:00:000 (i.e. 5 minutes). For the steady state calculations the time step in relation to the head increase is of importance because this determines the head drop increase that is applied per calculation step. In the current example, the head drop is increased by 4.5 m in 12 hours. This gives an increase of 0.03 m/time increment.

> The size of the head drop per time increment limits the maximum possible accuracy of the piping calculation. Other factors that influence the accuracy are the mesh size, the accuracy with which parameter values and boundary condition values are known and the accuracy of the schematisation. It is noted that the rate of increase does not influence the results for steady state calculations. In general the duration of a high water level would be much longer for a river dike than suggested in this example.

2. Set gravity to 10 and adapt MPicard to 1000. Do not adapt the other parameters under settings. Note that if the value of gravity is changed during the project, the corresponding hydraulic conductivity must be manually updated.

> MPicard controls the stepsize by which the pipe height is adjusted to achieve equilibrium. The stepsize is 100*D70/ MPicard. A larger stepsize (smaller MPicard) results in lower accuracy but faster calculation

3. In the Project window, go to River_dike_tutorial → D-Geo Flow Model → Input → Materials → Soils and right click on Soils and select "Add new soil". Click on the newly added soil.

4. In the Properties window Fill in the parameters for "Dike" stated in Table 6.1.

5. Repeat the previous step for all the other materials. It is recommended to use a different colour for each material. Note that it is also possible to right click a soil and copy the soil, useful for soils with the same parameters. The number will be assigned automatically in D-Geo Flow.

6. Set the water parameters as compressibility=0 m^2/N; density = 1000 kg/m^3; viscosity =0.001 Pa.s.

Table 6.1: Soil properties example 1, River dike

Name	Dike	Blanket	Top aquifer	Blanket polder side	Bottom aquifer
Porosity [-]	0.4	0.5	0.3	0.5	0.5
Compressibility [m^2/N]	0	0	0	0	0
Submerged particle density [kg/m^3]	n.a.	n.a.	1650	n.a.	n.a.
D70 [m]	n.a.	n.a.	0.0002	n.a.	n.a.
η [-]	n.a.	n.a.	0.25	n.a.	n.a.
θ [degrees]	n.a.	n.a.	37	n.a.	n.a.
Hydraulic Conductivity – X [m/day]	0.01	0.01	10	0.01	30
Hydraulic Conductivity – Y [m/day]	0.01	0.01	10	0.01	30

Table 6.2: Borehole data example 1, River dike

Borehole nr	1	2	3	4	5	6	7	8	9	10	11	12
X Position	0	6	10	20	32.5	37.5	48	58.25	60	63.75	67.5	100
Layer 1	Aquifer top	Aquifer top	Blanket	Blanket	Dike	Dike	Blanket	Blanket	Aquifer top	Aquifer top	Blanket polderside	Blanket polderside
Top level Layer 1	-2	-2	0	0	5	5	0	0	-2	-2	0	0
Layer 2	Aquifer bottom	Aquifer bottom	Aquifer top	Aquifer top	Blanket	Blanket	Aquifer top	Aquifer top	Aquifer bottom	Aquifer bottom	Aquifer top	Aquifer top
Top level Layer 2	-12	-12	-2	-2	0	0	-2	-2	-12	-12	-2	-2
Layer 3	-	-	Aquifer bottom	Aquifer bottom	Aquifer top	Aquifer top	Aquifer bottom	Aquifer bottom	-	-	Aquifer bottom	Aquifer bottom
Top level Layer 3	-	-	-12	-12	-2	-2	-12	-12	-	-	-12	-12
Layer 4	-	-	-	-	Aquifer bottom	Aquifer bottom	-	-	-	-	-	-
Top level Layer 4	-	-	-	-	-12	-12	-	-	-	-	-	-
Bottom level	-20	-20	-20	-20	-20	-20	-20	-20	-20	-20	-20	-20

7. In the project window, go to River_dike_tutorial → D-Geo Flow Model → Input → Boreholes and right click on Boreholes and select "Add new borehole". Double click on the newly added borehole.

8. For the newly added borehole, in the borehole window, keep the X position at 0 and set the Top level at -2 m.

9. In the borehole window, hover your mouse over the soil, a dropdown menu appears and select Aquifer top. Set the bottom level at -12 m.

10. Right click on the soil layer and select: "add layer", in the dropdown menu, select Aquifer bottom for the second layer. Set the bottom level at -20m. Repeat Step 7 to Step 10 for the remaining boreholes, stated in Table 6.2. Note to always double click on the newly added borehole to view this borehole. Note it is also possible to add boreholes by right clicking on a borehole and copying this borehole.

11. When all boreholes are created, in the project window, right-click on Boreholes and select "create cross-section".

12. In the project window, double click on D-Geo Flow model to view the geometry.

13. In the project window, go to River_dike_tutorial → D-Geo Flow Model → Input → Boundary Conditions. Right click on Boundary Conditions and select Add new General Submerging boundary condition.

14. In the Project window click on the newly added boundary condition and change the name to "Riverside Boundary" in the properties window. If preferred, the line colour of the boundary condition can be changed in the properties window as well.

15. In the Project window, expand the Riverside Boundary and double click on Head. Select "Table and Chart (single function)".

16. Fill in the values as stated in Table 6.3.

17. Repeat Step 13 and add the boundary condition for the polder level with values stated in Table 6.4.

18. Create a seepage boundary, 'Polderside Boundary', and a No-Flow boundary condition, 'No Flow'.

Table 6.3: Riverside head boundary condition

Time [yyyy/MM/dd HH:mm:ss]	Head [m]
2017-03-10 00:00:00	2
2017-03-10 12:00:00	6.5
2017-03-11 00:00:00	2

Table 6.4: Polder level head boundary condition

Time [yyyy/MM/dd HH:mm:ss]	Head [m]
2017-03-10 00:00:00	0
2017-03-10 12:00:00	0
2017-03-11 00:00:00	2

19. When the Boundary conditions are defined the Boundary conditions can be assigned to the geometry. In the D-Geo Flow Model window, click on a line and assign the corresponding boundary condition in the Properties window. It may be useful to uncheck the box boreholes in the map window. Multiple lines can be selected at once by dragging a square around the lines or by pressing the ctrl key and selecting multiple lines. In Figure 6.1, it is shown how the boundary conditions should be assigned. Note that the boundary conditions are to be assigned on both the lines and the nodes. In order to assign a boundary condition to a specific node it is necessary to double click the node. When no boundary condition is assigned, the boundary is a no-flow boundary, i.e. a zero flux boundary. In this example the zero-flux boundary conditions are assigned manually.

20. When the Boundary conditions are assigned, the pipe can be defined. Go to River_dike_tutorial → D-Geo Flow Model → Input → Pipes. Right click on Pipes and select "add new pipe"

21. Click on the newly added Pipe. In the Properties window, change Start: X to 6; and End: X to 60; Y to -2.

22. When the complete model is defined, click on River_dike_tutorial → D-Geo Flow Model → Input → Grid. In the Properties window, change Coarseness to 2.5 and change Pipe Coarseness to 6. This implies that the mesh elements have a general size of 2.5 m and the elements at the pipe are refined by a factor 6.

23. Right click on Grid in the Project window and select Generate grid. Check the appearance of the grid and the grid refinement, there is no automatic grid quality control in the D-Geo Flow GUI.

24. In the Project window, right click on D-Geo Flow Model and select "Run model" to start the calculation.
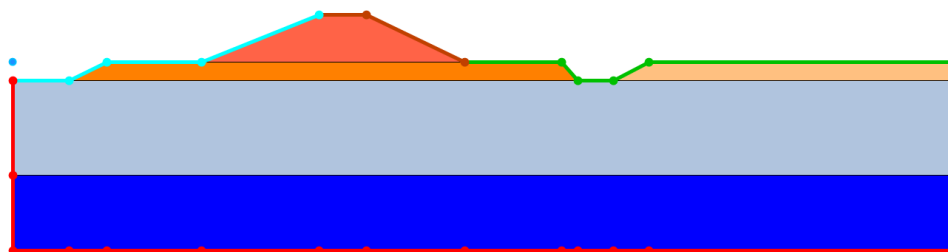
Figure 6.1: *Cross-section example River dike including boundary conditions. Red is No Flow boundary, light blue is River side boundary, purple is Polder side boundary and green is Polder level*

6.2.3 output

1. In the model view window, a visual presentation of the output is shown. In the Map window make sure that Map → D-Geo Flow Model → Results → Head, is selected. In the Time Navigator tab (bottom taskbar), click on play to see the development of the head in the domain over time.
2. In River_dike_tutorial → D-Geo Flow Model → Output → Pipe Results, time dependent results of the calculation are presented in tables and graphs. The tables can be viewed by double-clicking on the corresponding Pipe Result.
3. Double click on Equilibrium head drop, this shows the applied head drop and the corresponding pipe length up to the critical pipe length. The critical head drop is the last head drop before the pipe grows towards the other end of the dike, i.e. the second to last head drop in the table.
4. Double click on "Pipe Element Active" to view how the piping elements are activated over time. If the piping element is breached, the value for "Active" is 1. If the pipe element is not yet activated, its value is 2. Start the Time Navigator and search the first timestep where all the piping elements are breached; this should agree with the head just above the critical head.
5. Double click on "Head in Pipe" to view how the hydraulic head along the pipe path behaves over time. Navigate to one timestep before all the piping elements are activated. The maximum head at this timestep is the critical head.

6.3 Pilot Case 2: Sea dike

6.3.1 Start-up project

1. Repeat Step 1 to Step 4 from Example 1.
2. On the top taskbar click on File → Save As and safe your project as "Sea_dike_tutorial" in your preferred folder.

6.3.2 Input

1. Repeat input Step 1 to 4 from Example 1, however fill in the soil properties stated in Table 6.5. Note that the head increase per time step is now 0.035 m when a time step of 5 minutes is applied, since the head is increased to 5 m in 12 hours.

> Although the load is applied in a time-dependent manner, the calculation is still steady-state as long as the the soil and water are incompressible. The rate of increase is therefore irrelevant for the result, other than that it controls the step size by which the head is increased.

2. Set the water parameters as compressibility=0 m^2/N; density = 1000 kg/m^3; viscosity =0.001 Pa.s.
3. Repeat input Step 7-10 from Example 1, however fill in the borehole data stated in Table 6.6.
4. When all boreholes are created, in the project window, right-click on Boreholes and select "create cross-section".
5. In the project window, double click on D-Geo Flow model to view the geometry.
6. Analogous to input Step 13 to Step 16 from Example 1, create two head boundary conditions (Seaside boundary and Polder level boundary), a No-Flow boundary and a seepage boundary (Polder side boundary). Fill in the head values according to the tables below.

Table 6.5: Soil properties example 2, Sea dike

Name	Dike	Blanket	Blanket polder	Aquifer top	Aquifer bottom
Porosity [-]	0.4	0.4	0.4	0.3	0.3
Compressibility [m^2/N]	0	0	0	0	0
Submerged particle density [kg/m^3]	n.a.	n.a.	n.a.	1650	n.a.
D70 [m]	n.a.	n.a.	n.a.	0.0002	n.a.
η [-]	n.a.	n.a.	n.a.	0.25	n.a.
θ [degrees]	n.a.	n.a.	n.a.	37	n.a.
Hydraulic Conductivity – X [m/day]	0.01	0.01	0.01	10	30
Hydraulic Conductivity – Y [m/day]	0.01	0.01	0.01	10	30

Table 6.6: Sea dike Borehole data

Borehole nr	1	2	3	4	5	6	7	8	9	10	11
X Position	-20	4.8	8	45	47	77	100	101	105	106	150
Layer 1	Aquifer top	Aquifer top	Blanket	Dike	Dike	Blanket	Blanket	Aquifer top	Aquifer top	Blanket polder	Blanket polder
Top level Layer 1	0	0	4.1	14.1	14.1	4.1	4.1	0	0	4.1	4.1
Layer 2	Aquifer bottom	Aquifer bottom	Aquifer top	Blanket	Blanket	Aquifer top	Aquifer top	Aquifer bottom	Aquifer bottom	Aquifer top	Aquifer top
Top level Layer 2	-7.9	-7.9	0	4.1	4.1	0	0	-7.9	-7.9	-7.9	0
Layer 3	-	-	Aquifer bottom	Aquifer top	Aquifer top	Aquifer bottom	Aquifer bottom	-	-	Aquifer bottom	Aquifer bottom
Top level Layer 3	-	-	-7.9	0	0	-7.9	-7.9	-	-	-7.9	-7.9
Layer 4	-	-	-	Aquifer bottom	Aquifer bottom	-	-	-	-	-	-
Top level Layer 4	-	-	-	-7.9	-7.9	-	-	-	-	-	-
Bottom level	-15.9	-15.9	-15.9	-15.9	-15.9	-15.9	-15.9	-15.9	-15.9	-15.9	-15.9

Table 6.7: *Sea side head boundary condition*

Time [yyyy/MM/dd HH:mm:ss]	Head [m]
2017-03-10 00:00:00	0
2017-03-10 12:00:00	5
2017-03-11 00:00:00	0

Table 6.8: *Polder level head boundary condition*

Time [yyyy/MM/dd HH:mm:ss]	Head [m]
2017-03-10 00:00:00	0
2017-03-11 00:00:00	0

7. When the Boundary conditions are defined the Boundary conditions can be assigned to the geometry. In the D-Geo Flow Model window, click on a line and assign the corresponding boundary condition in the Properties window. Multiple lines can be selected at once by dragging a square around the lines or by pressing the ctrl key and selecting multiple lines. In Figure 6.2, it is shown how the boundary conditions should be assigned. Note that the boundary conditions are to be assigned on both the lines and the nodes.

8. When the Boundary conditions are assigned, the pipe can be defined. Go to River_dike_tutorial → D-Geo Flow Model → Input → Pipes. Right click on Pipes and select "add new pipe"

9. Click on the newly added Pipe. In the Properties window, change Start: X to -4.8; Y to 0 and End: X to 101.

10. When the complete model is defined, click on River_dike_tutorial → D-Geo Flow Model → Input → Grid. In the Properties window, change Coarseness to 2.5 and change Pipe Coarseness to 6.

11. Right click on Grid in the Project window and select Generate grid.

12. In the Project window, right click on D-Geo Flow Model and select "Run model" to start the calculation.
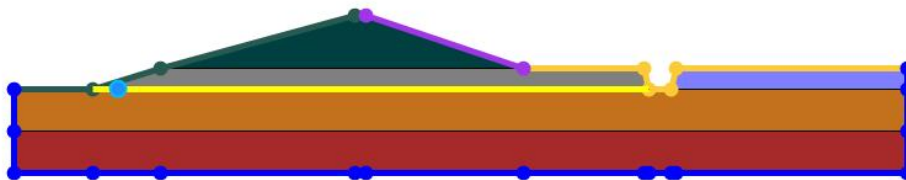
Figure 6.2: *Cross-section example 2 including boundary conditions (Blue: No Flow, Green: Seaside Head, Purple: Polderside, Yellow: Polder level).*

6.3.3 Output

Check the results in a similar manner as in example 1. In addition, look into the flow pattern and the pipe activation at different pipe lengths by checking the boxes 'Results', 'Head' and 'Pipe in the map. Note that the applied head was insufficient for the pipe to progress to the upstream side. To find the head drop at which the pipe will progress, the calculation can be repeated with an applied head drop of 3 to 8 m by applying the boundary conditions in Table 6.9.

Table 6.9: Adjusted sea side head boundary condition

Time [yyyy/MM/dd HH:mm:ss]	Head [m]
2017-03-10 00:00:00	3
2017-03-10 12:00:00	8
2017-03-11 00:00:00	3

7 Scientific background

7.1 Groundwater flow

7.2 Mathematical Model

The storage equation according to Richard's model follows from the mass balance equation, Darcy's law and empirical relations for water content and relative permeability as a function of pore pressure. This article adopts a Galerkin finite element discretization in space, in which the primary field variable is pore pressure $p\ (\mathrm{N/m^2})$ Hughes (2000); Belytschko *et al.* (2008). Pore pressures are considered to be positive in compression, with index notation being used, $x_i\ (\mathrm{m})$ denotes Cartesian coordinates in space, $t\ (\mathrm{s})$ denotes time, and $\Omega\ (\mathrm{m^2})$ represents the flow domain with $\Gamma\ (\mathrm{m})$ its boundary.

7.2.1 Subsurface Flow Equation

Flow through a partly saturated porous medium can be modeled considering conservation of mass and a generalization of Darcy's law Van Esch (2010), which can be written as

$$(\alpha + n\beta)\,S\frac{\partial p}{\partial t} + n\frac{dS}{dp}\frac{\partial p}{\partial t} + \frac{\partial q_i}{\partial x_i} = 0\ , \quad q_i = -\frac{k_r \kappa_{ij}}{\mu}\left(\frac{\partial p}{\partial x_j} - \rho^l g_j\right) \quad \text{on}\quad \Omega^p\ , \quad (7.1)$$

where $\alpha\ (\mathrm{m^2/N})$ is the compressibility of the soil skeleton, $\beta\ (\mathrm{m^2/N})$ represents the compressibility of the pore water, $n\ (-)$ denotes porosity and $S\ (-)$ expresses the degree of saturation of the liquid phase in the void space. The compressibility for elastic behavior of the soil skeleton can be written as $\alpha = 1/(\lambda + 2\nu)$, where $\lambda\ (\mathrm{N/m^2})$ and $\nu\ (\mathrm{N/m^2})$ denote Lamé's constants. Specific discharge $q_i\ (\mathrm{m/s})$ relates to relative permeability $k_r\ (-)$, intrinsic permeability $\kappa_{ij}\ (\mathrm{m^2})$, dynamic viscosity of the liquid phase $\mu\ (\mathrm{kg/ms})$ and its density ρ^l $(\mathrm{kg/m^3})$. For a y-coordinate pointing in the opposite direction than the gravitational vector g_y is $-9.81\,\mathrm{m/s^2}$. Equation (7.1) is known as the storage equation. Two types of boundary conditions complete the problem definition; Dirichlet conditions prescribe the pressure on parts of the boundary and Von Neumann boundary conditions prescribe the derivative of the pressure or flux on the boundary. These first-type and second-type boundary conditions read

$$p = \overline{p}\quad \text{on}\quad \Gamma_1^p\ , \quad q_i n_i = -\overline{q}\quad \text{on}\quad \Gamma_2^p\ , \quad q_i n_i = -s\quad \text{on}\quad \Gamma_3^p\ , \quad (7.2)$$

where Γ_1^p, Γ_2^p and Γ_3^p are disjoint parts of the boundary where the conditions apply, $n_i\ (-)$ denotes the outward pointing normal to the boundary, $\overline{p}\ (\mathrm{N/m^2})$ is the prescribed pressure, $\overline{q}$ $(\mathrm{m/s})$ denotes the volumetric water flux over the boundary into the flow domain as a source term, and $s\ (\mathrm{m/s})$ expresses a source term that will be used for coupling the subsurface flow equation to the flow equation for piping.

A weak form of equation (7.1) follows from multiplication by weighting functions $N_a\ (-)$, which are attached to nodes of a finite element mesh and integration over the flow domain. The first weak form is given by

$$\int_{\Omega^p} N_a\,(\alpha + n\beta)\,S\frac{\partial \hat{p}}{\partial t}d\Omega + \int_{\Omega^p} N_a n\frac{dS}{dp}\frac{\partial \hat{p}}{\partial t}d\Omega = \int_{\Omega^p} N_a \frac{\partial}{\partial x_i}\left[\frac{k_r \kappa_{ij}}{\mu}\left(\frac{\partial \hat{p}}{\partial x_j} - \rho^l g_j\right)\right]d\Omega\ ,$$
$$(7.3)$$

where the caret above a symbol implies that the variable is an approximation. Application of

Green's theorem to the term on the right hand side of equation (7.3) leads to

$$
\int_{\Omega^p} N_a \left(\alpha + n\beta\right) S \frac{\partial \hat{p}}{\partial t} d\Omega + \int_{\Omega^p} N_a n \frac{dS}{dp} \frac{\partial \hat{p}}{\partial t} d\Omega =
$$
$$
- \int_{\Omega^p} \frac{k_r \kappa_{ij}}{\mu} \frac{\partial N_a}{\partial x_i} \left(\frac{\partial \hat{p}}{\partial x_j} - \rho^l g_j \right) d\Omega + \int_{\Gamma^p_{2 \cup 3}} N_a \frac{k_r \kappa_{ij}}{\mu} \left(\frac{\partial \hat{p}}{\partial x_j} - \rho^l g_j \right) n_i d\Gamma .
$$

$$(7.4)$$

Galerkin weighting and substitution of the interpolation functions gives

$$
\int_{\Omega^p} N_a N_b \left(\alpha + n\beta\right) S \frac{dp_b}{dt} d\Omega + \int_{\Omega^p} N_a N_b n \frac{dS}{dp} \frac{dp_b}{dt} d\Omega = - \int_{\Omega^p} \frac{k_r \kappa_{ij}}{\mu} \frac{\partial N_a}{\partial x_i} \frac{\partial N_b}{\partial x_j} p_b d\Omega
$$
$$
+ \int_{\Omega^p} \frac{k_r \kappa_{ij}}{\mu} \frac{\partial N_a}{\partial x_i} \rho^l g_j d\Omega + \int_{\Gamma^p_2} N_a N_b \bar{q}_b d\Gamma + \int_{\Gamma^p_3} N_a N_b s_b d\Gamma , \quad (7.5)
$$

after the Neumann boundary conditions were incorporated. Prescribed boundary fluxes are given at the nodal points. Implicit time integration yields

$$
\int_{\Omega^p} N_a N_b \left(\alpha + n\beta\right) S^{k+1} \left(p_b^{k+1} - p_b^k \right) d\Omega + \int_{\Omega^p} N_a N_b n \left(\frac{dS}{dp} \right)^{k+1} \left(p_b^{k+1} - p_b^k \right) d\Omega =
$$
$$
- \Delta t \int_{\Omega^p} \frac{k_r^{k+1} \kappa_{ij}}{\mu} \frac{\partial N_a}{\partial x_i} \frac{\partial N_b}{\partial x_j} p_b^{k+1} d\Omega + \Delta t \int_{\Omega^p} \frac{k_r^{k+1} \kappa_{ij}}{\mu} \frac{\partial N_a}{\partial x_i} \rho^l g_j d\Omega
$$
$$
+ \Delta t \int_{\Gamma^p_2} N_a N_b \bar{q}_b^{k+1} + \Delta t \int_{\Gamma^p_3} N_a N_b s_b^{k+1} d\Gamma , \quad (7.6)
$$

where Δt is the time step size, k denotes the current time step and $k + 1$ indicates the new step.

7.2.2 Flow Equation for Piping

Modeling of piping follows from the application of conservation of mass and considering Poiseuille flow Huyakorn and Pinder (1983) through a horizontal erosion slit and can be written as

$$
\frac{dq}{dx} + s = 0 , \quad q = - \frac{a^3}{12\mu} \frac{dp}{dx} \quad \text{on} \quad \Omega^c , \tag{7.7}
$$

where $a \; (\mathrm{m})$ denotes the height of the channel and s expresses a sink term. Here the extension of the pipe flow domain $\Omega^c \; (\mathrm{m})$ corresponds to the boundary of the subsurface flow domain $\Gamma^p \; (\mathrm{m})$ as equation (7.7) holds for a one-dimensional flow domain. Dirichlet conditions hold on the inflow and outflow point of the piping domain. Boundary conditions for the piping domain are given by

$$
p = \bar{p} \quad \text{on} \quad \Gamma^c_1 . \tag{7.8}
$$

The condition of limit equilibrium for the horizontally oriented erosion channel Sellmeijer (1988) reads

$$
a \frac{dp}{dx} = \frac{\pi}{3} \left(\rho^s - \rho^l \right) g d_{70} \eta \tan \vartheta . \tag{7.9}
$$

The height of the erosion channel varies in between 0 and 3 grains in an experimental setup, 0 to 10 grains in a field test and 0 to 30 grains for an embankment.

Discretization of equation (7.7) follows the same steps as the discretization of equation (7.1), and the first weak form reads

$$\int_{\Omega^c} N_a \frac{d}{dx}\left(\frac{\kappa_p}{\mu}\frac{d\hat{p}}{dx}\right)d\Omega = \int_{\Omega^c} N_a s d\Omega \;, \tag{7.10}$$

where the equivalent intrinsic permeability $\kappa_p\;(\mathrm{m}^3)$ has a scalar value that is given by $\kappa_p = a^3/12$. The second weak form is given by

$$\int_{\Omega^c} \frac{\kappa_p}{\mu}\frac{dN_a}{dx}\frac{d\hat{p}}{dx}d\Omega = -\int_{\Omega^c} N_a s d\Omega \;, \tag{7.11}$$

and the final form follows from the substitution of interpolation functions

$$\int_{\Omega^c} \frac{\kappa_p^{k+1}}{\mu}\frac{dN_a}{dx}\frac{dN_b}{dx}p_b^{k+1}d\Omega = -\int_{\Omega^c} N_a N_b s_b^{k+1}d\Omega \;. \tag{7.12}$$

The condition of limit equilibrium for a horizontally oriented erosion channel is evaluated at the integration points and is given by

$$\kappa_p^{k+1} = \tfrac{1}{12}\left(a^{k+1}\right)^3, \quad a^{k+1}\frac{dN_b}{dx}p_b^{k+1} = \frac{\pi}{3}\left(\rho^s - \rho^l\right)gd_{70}\eta\tan\vartheta \;. \tag{7.13}$$

7.2.3 Coupled Flow Equation

The sink term in equation (7.12), where s is positive for flow out of the channel, provides a source term in equation (7.6), where s is positive for flow into the subsurface, and vice versa. Linear one-dimensional elements are used for solving equation (7.12), whereas linear triangular elements discretize the flow domain attached to equation (7.6). Combining both equations yields

$$\int_{\Omega^p} N_a N_b\left(\alpha + n\beta\right)S^{k+1}\left(p_b^{k+1} - p_b^k\right)d\Omega + \int_{\Omega^p} N_a N_b n\left(\frac{dS}{dp}\right)^{k+1}\left(p_b^{k+1} - p_b^k\right)d\Omega =$$
$$-\Delta t\int_{\Omega^p}\frac{k_r^{k+1}\kappa_{ij}}{\mu}\frac{\partial N_a}{\partial x_i}\frac{\partial N_b}{\partial x_j}p_b^{k+1}d\Omega + \Delta t\int_{\Omega^p}\frac{k_r^{k+1}\kappa_{ij}}{\mu}\frac{\partial N_a}{\partial x_i}\rho^l g_j d\Omega$$
$$-\Delta t\int_{\Gamma_3^p}\frac{\kappa_p^{k+1}}{\mu}\frac{\partial N_a}{\partial x_i}\frac{\partial N_b}{\partial x_j}p_b^{k+1}d\Omega + \Delta t\int_{\Gamma_2^p}N_a N_b \overline{q}_b^{k+1} \;. \tag{7.14}$$

The interface elements align along the horizontal axis. Picard iteration over κ_r^{k+1} and S^{k+1} are used to resolve the non-linearities in the system of equations. Nonlinearities in κ_p^{k+1} are resolved sequentially. For a given trial channel height, equation (7.13) provides the equivalent intrinsic permeability, which is attached to the relevant interface element. The product of channel height and pressure gradient are then compared with the equilibrium condition given by the second part of expression (7.13).

More about validating the DgFlow kernel can be found in Taccari (2016)

7.3 More about piping

Figure 7.1 provides the sequential procedure graphically for a sample problem that includes two interface elements that are in series. The height of the channel is presented as a function of hydraulic head gradient. Hydraulic head $h\;(\mathrm{m})$ is presented as $h = p/(\rho^l g) + y$, where $y\;(\mathrm{m})$ is the elevation level. The dark blue line represents the limit equilibrium condition; below this line stable grain conditions apply and above the equilibrium line unstable conditions hold.
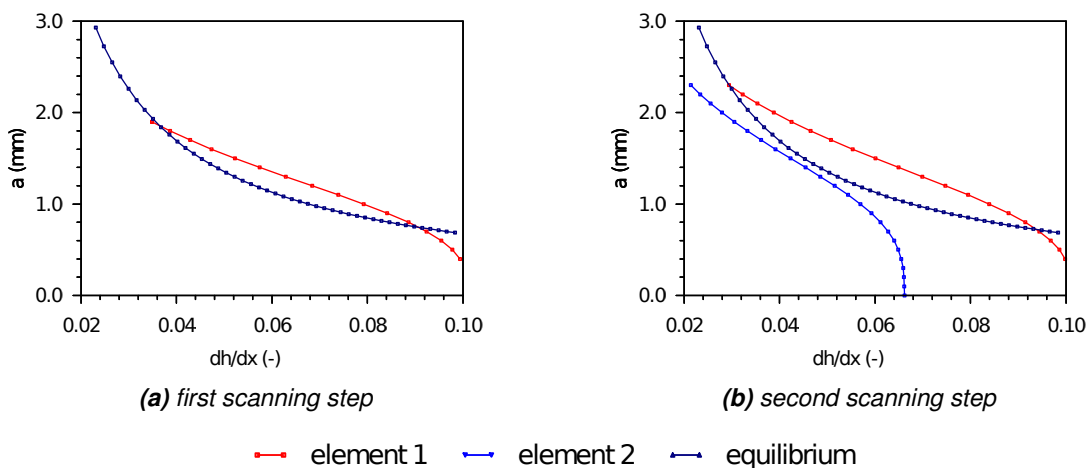
(a) *first scanning step* **(b)** *second scanning step*

element 1   element 2   equilibrium

Figure 7.1: *Activation algorithm.*

The calculation process starts by checking the interface element located to the downstream side of the channel (element 1) next to the exit point. For a thickness of 0 mm the head gradient is due to the flow process through the subsurface. Increasing the thickness of the element increases the effective permeability of the element and decreases the gradient over the element leading to the red line in Figure 7.1 (a). The height of the channel is increased stepwise until the condition becomes unstable. This unstable situation is associated with the first (right) point where the red line intersects the equilibrium line. At this point the element becomes active for the given head difference over the structure; a small increase in the channel height pushes the element in the unstable zone. The permeability of the element is increased until the element becomes stable again. The element thickness for which equilibrium holds is obtained by means of a bisection algorithm. This second (left) intersection point denotes a stable situation. If the height of the channel increases above this point, the element enters the stable zone.

The procedure continues by checking the element on the upstream side (element 2) of the first element (element 1). The plot on the right of Figure 7.1 illustrates this process. The height is increased sequentially until the line enters the unstable zone and the second intersection point is found, or a threshold value for the maximum channel height has been reached indicating that the element will not be activated. For the depicted situation, the applied head difference over the structure will not activate the second element. If equilibrium is not satisfied for the trial element however, then the height of the already activated elements is modified accordingly until equilibrium is satisfied within the entire pipe domain.

More about validating the piping module in D-Geo Flow can be found in Rosenbrand and Van Beek (2017)

7.4 Default and recommended settings

Table 7.1: *material parameter ranges*

parameter	symbol	minimum	default	maximum	unit
particle diameter	D_{70}	$1.00 \cdot 10^{-4}$	$2.00 \cdot 10^{-4}$	$8.00 \cdot 10^{-4}$	m
density of the grains	ρ^g	$2.40 \cdot 10^3$	$2.65 \cdot 10^3$	$2.80 \cdot 10^3$	$\mathrm{kg/m^3}$
White's constant	η	0.20	0.25	0.30	$-$
bedding angle	ϑ	30	37	50	deg
intrinsic permeability	κ	10^{-16}	$1.157 \cdot 10^{-11}$	10^{-10}	$\mathrm{m^2}$
pore water density	ρ^l	10^3	10^3	10^3	$\mathrm{kg/m^3}$
dynamic viscosity water	μ^l	$8.0 \cdot 10^{-4}$	$1.3 \cdot 10^{-3}$	$2.0 \cdot 10^{-3}$	$\mathrm{Ns/m^2}$
gravitational acceleration	g	9.00	9.81	11.00	$\mathrm{m/s^2}$
porosity	n	0.2	0.36	0.9	$-$
minimal saturation	S_r	0.0	0.00	1.0	$-$
maximal saturation	S_s	0.0	0.36	1.0	$-$
Genuchten shape factor	g_a	0.3	2.24	2.5	$1/\mathrm{m}$
Genuchten shape factor	g_n	1.0	2.286	2.5	$-$
Genuchten shape factor	g_l	-6.0	0.00	1.0	$-$
compressibility soil	α	10^{-10}	$1.0 \cdot 10^{-7}$	10^{-5}	$\mathrm{m^2/N}$
compressibility water	β	10^{-10}	$5.0 \cdot 10^{-10}$	10^{-5}	$\mathrm{m^2/N}$

References

Belytschko, T., W. K. Liu and B. Moran, 2008. *Nonlinear Finite Elements for Continua and Structures*. Wiley.

Hughes, T. J. R., 2000. *The Finite Element Method, Linear Static and Dynamic Finite Element Analysis*. Dover Publications.

Huyakorn, P. S. and G. F. Pinder, 1983. *Computational Methods in Subsurface Flow*. Academic Press.

Rosenbrand, E. and V. van Beek, 2017. *Validation piping module in D-Geo Flow GUI update, Analysis with steady state and transient flow; D-Geo Flow GUI version spring 2017 (available on request)*. Tech. rep., Deltares.

Sellmeijer, J. B., 1988. *On the mechanism of piping under impervious structures*. Ph.D. thesis, Delft University of Technology.

Taccari, M., 2016. *Validation DgFlow. Memo 24th of October 2016 project 1221071*. Tech. rep., Deltares.

Van Esch, J. M., 2010. *Adaptive Multiscale Finite Element Method for Subsurface Flow Simulation*. Ph.D. thesis, Delft University of Technology.
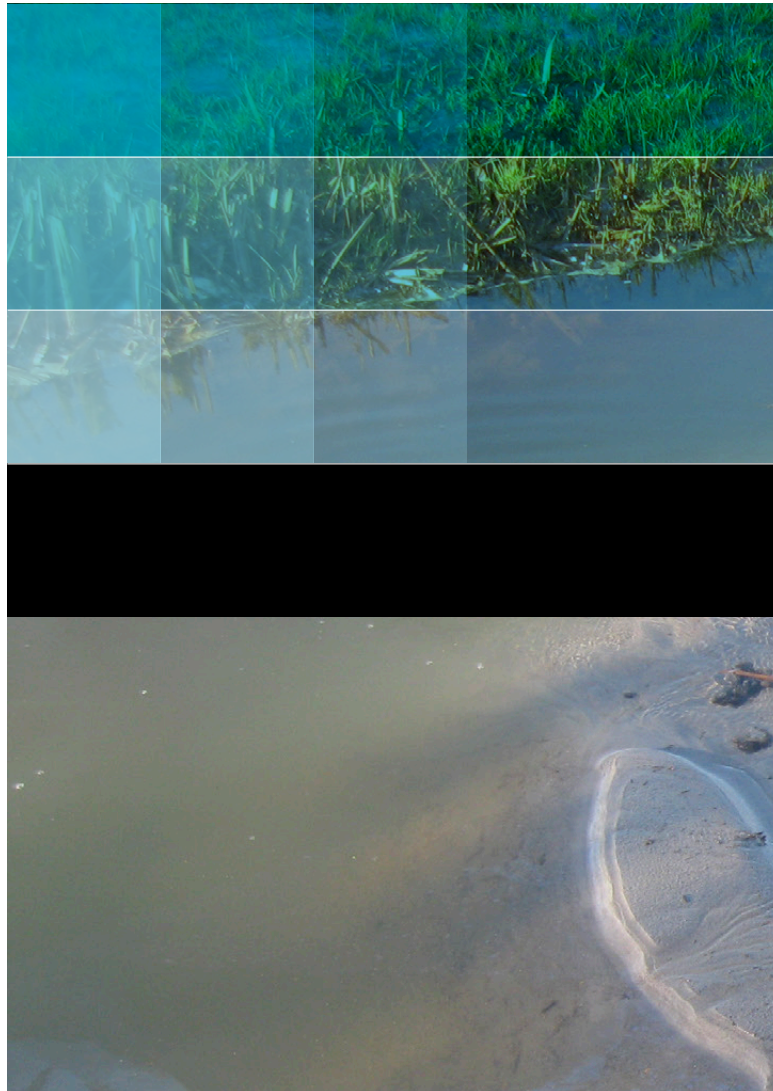
Photo by: Waterschap Rivierenland

Deltares **systems**