NTNU
Norwegian University of
Science and Technology

Numerical simulation of sediment flushing in reservoirs

Alexander Anatol Ermilov

Norwegian University of Science and Technology

Faculty of Engineering

Department of Civil and Environmental Engineering

Numerical simulation of sediment flushing in reservoirs

Master thesis

2017/18. I. semester

Author:

Alexander Anatol Ermilov

Supervisors:

Dr. Nils Rüther

Dr. Sándor Baranya

Declaration

I, Alexander Anatol Ermilov, MSc civil engineering student of Budapest University of Technology and Economics, hereby declare that I wrote this thesis without using any forbidden means of help. Every part that is based on the work of other people is marked with the source in the text and in the *References* part.

Abstract

As the awareness and need of using sustainable energy sources are increasing through the global society, it makes the developers and investors to take bigger steps towards improving these technologies and to overcome difficulties that come with them. Hydropower development is one of these possible ways, with a huge potential. However, with huge potential come lot of challenges. From this point of view, a core problem can be the sedimentation processes. Experience shows sediment is a factor that needs to be taken into consideration when planning or maintaining power plants. We have to consider the impact the designed structure can make on the sedimentology of the downstream or even plan flushing practices for the maintenance period. Hence, the knowledge of these processes is inevitable.

However, describing sedimentation is not always easy. For example, there are many formulas for describing bedload transport, mainly based on empirical equations, but there is no generally used and accepted one. Usually, it is needed to use more of them and give a range of the magnitude of the expected transport. It is seen that carrying out further experiments and measurements can lead to deepening the knowledge of the processes and through this, helping to improve the efficiency of hydropower development.

At the same time as we progress with better understanding all of the above mentioned aspects, our computers and their efficiency are also improving, making them a useful tool to aid us during the process. There are plenty of numerical tools nowadays to choose from and simulate the area of interest. However, if we come to talk about sedimentation, one may find flexibility in the open-source tools, where they can make modifications according to what the specific situation requires. Moreover, if we just think about the mentioned wide range of formulas, an open-source tool can be handy to add new ones, compare them easily with others, find the common aspects and figure out something uniformal.

In my thesis, I used the free and open-source TELEMAC-MASCARET software package and tested its adaptability to a sediment flushing scenario, which was carried out in the laboratory of the Department of Civil and Environmental Engineering at the Norwegian University of Science and Technology in Trondheim. The goal was to show if it can be used in this case, and if yes, then how well it represents the result of the laboratory physical model.

Table of contents

I. Introduction

1. The problem of sediment yield in reservoirs

According to earlier studies, since we disrupt the natural course of the sediment transport through rivers by building dams, we cause a deposition in the upstream of the reservoir (*Kondolf et al., 2014*). This leads to losing storage area, the amount of which is calculated to be around 0.5% loss per year, globally (*White, 2001*). Hence, as *Kondolf et al., 2014; Morris and Fan, 1998; Shen, 1999.* all point it out, sediment deposition can reduce the lifespan of these structures and also decrease the function of the reservoir for, for example, hydropower development or flood control, resulting in economic loss. This sedimentation usually follows the same rule. The coarse material, which was transported as bedload up until that point, is deposited at the entrance of the reservoir, where the flow pattern changes, and it takes up a delta formation. Meanwhile, the suspended load travels into the reservoir and settles down uniformly (*Scheuerlein, 1991*). For this problem, there are three generally used methods, such as: limiting the amount of incoming sediment yield, limiting the deposition, and removing the deposited sediment from the reservoir (*Healey et al., 2015*). For *minimizing siltation*, the usual methods are: constructing *sediment bypass* structures (i.e. canals, pressurized pipelines, or tunnels), *sluicing* (*sediment pass-through*) and *density current venting*. As for removing deposits: *drawdown flushing*, *pressurized flushing* and *dredging* are the usual solutions. All these techniques have their own benefits and drawbacks.

As the hydropower constructions of dams affects the sedimentation processes, it has an impact on both the physical and the biological environment of the downstream. For instance, some of the above mentioned solutions create high suspended sediment concentrations, which can lead to lethal consequences for the fish population (*Wilber et al., 2001; Robertson et al., 2006*). The magnitude of the impact depends on the time between the flushings. The more time is spent between the flushings, higher is the concentration that is released, so the impact is bigger. However, frequent flushings are proven to be more expensive.

Incoming sediment particles are also causing erosion in the turbines of power plants, reducing their efficiency (*Thapa, 2004; Neopane, 2010; Eltvik, 2013*). In this topic, only a few studies have been conducted so far.

As a response to the problems above, Norway decided to start a project, called "*Sustainable design and operation of hydro power plants exposed to high sediment yield – SEDIPASS*". Since the country has 20 % of the hydropower resources of Europe, hydropower plays an important part in the Norwegian economy and they are also a major investor in the global energy market. The project has several work packages with the over-all goal to "*develop knowledge towards the improved design and operation of sustainable hydro power plants exposed to high sediment yield*" (*Guerrero and Rüther, 2014*). For the secondary objectives, they intend to develop knowledge on: a more reliable, less time and money consuming *method of measuring sediment concentration and grain sizes*; *improved physical models* with sediment transport; quantifying the magnitude of *erosion on turbines* due to sediment; quantifying *sediment loads due to* different *flushing/dredging operations*; *develop industrial guidelines* based on the results. The Norwegian University of Science and Technology plays a major role in this project.

The physical model, which is the base of the numerical model in this thesis, was also created for this SediPASS project and they are carrying out the measurements in accordance with its objectives.

2. Numerical tools for hydraulic and sedimentation engineering

Using physical models for solving and understanding hydraulic and sedimentation problems is a quite usual method. However, it can be time-consuming and expensive. Also, engineers may face the problem of scaling. For example, as we increase the scale-down, the cohesive forces will play more significant role in our physical model (*Nils Olsen, 1999)*. These physical models are based on dynamic similarity, but it is not possible to model all force ratios accurately. The scale-effect must be taken into consideration, when the results are being used. Models with artificial granulates can provide, for example, similar grain Reynolds number, but the relative sediment density requirements or fall speed may differ. Even though, lightweight models proved to give good, qualitative results for structure-sediment interactions. However, scaling, ordering and producing these materials, or even the physical model of the structure, takes time and financial resources. Also, a mistake, or a bigger change in the model can further increase these expenditures. Hence, numerical models can be helpful for aiding or replacing them. A well-built and calibrated computational fluid dynamics (CFD) model can speed up analysing different scenarios and cost less at the same time.

Nowadays, there are plenty of CFD models that can be used for hydraulic and sediment purposes. Hereby, a short introduction of some of them is given.

ANSYS FLUENT is a powerful CFD tool, based on the finite volume method with a wide range of applicability (laminar-turbulent, incompressible-compressible, steady-transient), including sediment modelling as well. For instance, it can be used for simulating the interaction of the moving turbines and the incoming particles, to show the amount of erosion, as mathematical models for transport phenomena is be combined with modelling complex geometries (*ANSYS FLUENT Theoretical Guide, 2009*). Of course, the powerful tool and wide range of service causes it to be relatively expensive (*Nils Olsen, 1999*), but there is a free version for students and discounts for academic purposes.

SSIIM (Sediment Simulation In Intakes with Multiblock option) is a 3D model for free surface flows, using the finite volume method. It is developed at the Norwegian University of Science and Technology. It is free and its main purpose is river hydraulic and sedimentation modelling. The software is using the k-epsilon turbulence model and it can be used with unstructured grid as well. Since it is developed by an academic institution, user support is not available. Only the released manuals and documentation provide help. (*Nils Olsen, 2014*)

MIKE 3 is also a commercial 3D modelling tool, used for marine purposes, to model free surface flows and associated sediment or water quality processes, however, it may also be applied for inland surface waters (e.g. lakes). It is using the finite volume method like the previously mentioned ones. (*Mike 21/3 User Guide, 2017*)

TELEMAC-MASCARET modelling system is an integrated tool for free surface flows, based on the finite element method. Coupled together, the modules TELEMAC-3D and SISYPHE are used for hydraulic and sedimentation modelling. It has applications for both river and maritime purposes. It started out as a commercial modelling tool, but now it is free and available as open source. (*Sébastien Bourban, 2014*)

In this thesis the latter one was used, taking advantage of its open source and the fact that it is free to use. The goal was to test the applicability of TELEMAC-3D and Sisyphe in a sediment flushing scenario, by comparing the results to the physical model.

3. TELEMAC cases for sediment processes

TELEMAC has been used in numerous hydro-sedimentary studies so far. Shortly, some of the latest ones, which were presented at the XXIIIrd TELEMAC-MASCARET User Conference in 2016, are introduced below.

TELEMAC was used in the case study of Champagneux run-of-river dam on Rhône River, France. The goal was to use a flushing event to calibrate a 3D sediment transport model through the reservoir (*Alliau et al., 2016*). The flushing was done on the French upper Rhône River, through the Genissiat dam. The concentration that can reach the downstream through this dam is limited in order to avoid harmful effects on the fluvial environment. As the reservoir of Champagneux is located on the downstream, it is concerned by the operations conducted at Genissiat. The sediment is composed of potentially cohesive fine sediment and lead to an aggradation of the bed. They were looking for the hydrodynamic conditions that are necessary to set the deposits in motion and if a satisfactory state of morphological balance can be reached or not. Telemac-3D was used for the hydraulic computations, Sisyphe for the bed load and Sedi-3d for suspended sediment modelling. The numerical domain was 4 km long. The sub-surface concentration results showed accurate correlation between calculations and measurements. However, the critical erosion shear stress, the recent deposit and the bed evolution calibration proved to be difficult because of time variation and uncertainties of sediment parameters. After this, they set the goal of improving the model to make it possible to handle time-dependant vertical concentration profiles and settling velocity as a function of concentration.

Another study was conducted to compare measurements of cross-sectional variation of bed load transport with simulations (*Kopmann et al., 2016*). They used the measured data of an earlier study, which had more than 10,000 bed load samples from the Lower-Rhine, and complemented it with simulations, to get a satisfactory statistical dataset. The domain was 46.5 km long and the simulation period was 11.5 years. First, they validated the model by comparing the simulated effective bed load width and the centre of mass of the transport to the measured ones. The agreement was satisfactory. After that, they proceeded to carry out simulations with artificial bed load supplies and came to the conclusion that the coarse bed load supply decreases the effective transport width, but increases the cross-sectional variation of the transport.

The last case mentioned here, was aiming for exploring the consolidation process of the Río de la Plata estuary and implementing a high resolution 3D wave-current-sediment transport model to simulate the flow field and transport, focusing on the area of the Montevideo Bay (*Santoro et al., 2016*). They successfully calibrated the consolidation model and got good agreements for the vertical bed density profiles. This way, they were able to have spatial variability on the erosion parameters. Where the erosion was stronger, the top layer had higher sediment concentration and higher critical shear stress for erosion. The hydrodynamic and sediment model was also successfully implemented. The sensitivity analysis showed the importance of the wind drag coefficient, through its effect on the salinity distribution and currents.

4. Objectives of the thesis

The general goal of this thesis was to numerically recreate a sediment flushing scenario of a given physical model, using the TELEMAC-MASCARET modelling system. As it is free and open source, it has been used in various cases by users with success. It was found desirable, to test it and its flexibility in a sediment flushing scenario as well, hence the topic of the thesis.

The tasks first of all, composed of finding out if it was possible to model the scenario, and if yes, then up until what point was Telemac applicable. The main question was how a pressurized flow through a sluice gate could be modelled with Telemac, which is a free surface flow model originally, and what sacrifices, simplifications needed to be done.

After that, evaluation of the simulation results needed to be carried out, with describing possible future improvements of the numerical model.

All these were done after getting familiar with using the Telemac-3D flow-, and Sisyphe sediment modules through a test case.

II. The numerical model

1. Theoretical background of the numerical tool

In this thesis, the free and open source Telemac system was used. Specifically, the *Telemac-3D* and *Sisyphe* modules. The main difference from other competent modelling systems (e.g. Mike 3) is the use of the *Finite Element Method* (FEM), which provides its flexibility (*Villaret et al., 2011*). However, the user has the option to choose to use the Finite Volume Method if necessary. It is possible to do parallelisation with domain composition, and the partitioning is conducted without any overlapping. The codes and programs are in Fortran 90 and can be run on Windows, Linux and Unix as well. Thanks to the optimised FEM numerical schemes, Telemac can be used from laboratory experiment scale (*see II. 2.*) to river and marine applications (*see I. 3.*). Lately, it was also used in Earth-scale for simulating tsunami events, global tides, internal tides and water surges driven by storm events (further information can be found on the homepage of Telemac). The modelling system consists of different modules and the users have to internally couple those they wish to use for the case of interest. All of them are based on unstructured grids made up of triangles. In the following, a brief introduction of the modules used for this thesis is given, emphasizing only their aspects that had to be applied in the current case.

1.1. Telemac-3D

Telemac-3D is the 3D flow module of the Telemac system, responsible for the 3D hydrodynamic computations. It solves the Reynolds-averaged Navier-Stokes (RANS) equations (with hydrostatic pressure hypothesis or with the non-hydrostatic hypothesis) and the transport equation (where the tracers are categorized into active and passive groups). The results at every time step are the velocity components in the three directions (U, V, W) and the concentration of the tracer(s) at every point of the 3D mesh, also the water depth in the points of the 2D mesh. The mesh movement is taken into consideration through the σ-transformation. The users can define if they want to have evenly spaced horizontal levels, or levels distributed according to given proportions, or levels with specified elevations or the mixture of all these methods. Telemac-3D duplicates the 2D mesh along the vertical domain and in the end, we get the 3D mesh, made up of prisms.

By default, the following assumptions are used. 3D Navier-Stokes equations with time-varying free surface, incompressible fluid, hydrostatic pressure hypothesis and Boussinesq approximation for the momentum. Hence, we get the following 3D-equations:

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} = 0$$

$$\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} + W\frac{\partial U}{\partial z} = -g\frac{\partial Z_s}{\partial x} + v\,\Delta(U) + F_x$$

$$\frac{\partial V}{\partial t} + U\frac{\partial V}{\partial x} + V\frac{\partial V}{\partial y} + W\frac{\partial V}{\partial z} = -g\frac{\partial Z_s}{\partial y} + v\,\Delta(V) + F_y$$

$$p = p_{atm} + \rho_0 g(Z_s - z) + \rho_0 g \int_z^{Z_s} \frac{\Delta \rho}{\rho_0} dz'$$

$$\frac{\partial T}{\partial t} + U\frac{\partial T}{\partial x} + V\frac{\partial T}{\partial y} + W\frac{\partial T}{\partial z} = \text{div}(v\,\text{grad}\,T) + Q$$

where:

    U, V, W [m/s]: 3D velocity components

    h [m] : water depth

    Z_S [m]: free surface elevation

    F_x, F_y [m/s^2]: source terms (wind, Coriolis force, bottom friction etc.)

    g [m/s^2]: gravitational acceleration

    x, y, z [m]: space components

    t [s]: time

    v [m^2/s]: kinematic viscosity or tracer diffusion coefficients

    p : pressure

    p_{atm} : atmospheric pressure

    ρ_0 : reference density

    $\Delta \rho$: change of density around the reference density

    T (g/L, °C, etc.) : active/passive tracers

    Q : tracer source or sink

    h, U, V, W and T are the unknown variables in the equations.

In case we choose the non-hydrostatic version, then the equations above are modified:

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} + \frac{\partial W}{\partial z} = 0$$

$$\frac{\partial U}{\partial t} + U\frac{\partial U}{\partial x} + V\frac{\partial U}{\partial y} + W\frac{\partial U}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \upsilon\,\Delta(U) + F_x$$

$$\frac{\partial V}{\partial t} + U\frac{\partial V}{\partial x} + V\frac{\partial V}{\partial y} + W\frac{\partial V}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \upsilon\,\Delta(V) + F_y$$

$$\frac{\partial W}{\partial t} + U\frac{\partial W}{\partial x} + V\frac{\partial W}{\partial y} + W\frac{\partial W}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial z} - g + \upsilon\,\Delta(W) + F_z$$

$$p = p_{atm} + \rho_0 g(Z_S - z) + \rho_0 g \int_z^{Z_S} \frac{\Delta\rho}{\rho_0} dz + p_d$$

As it can be seen, the pressure in this case is composed of the hydrostatic pressure and an additional dynamic pressure term.

For the turbulent viscosity, it is possible to choose between using the constant viscosity, Smagorinski, mixing length, k-ε or the k-ω methods.

1.2. Sisyphe

Sisyphe is the module responsible for sediment transport and bed evolution computations. The module groups the sediment processes into three groups: bedload, suspended sediment or total load. It can be used for non-cohesive (uniform or non-uniform) and cohesive sediment as well. Modelling sand-mud mixtures and vertical stratification is also possible.

After coupling, the flow module (Telemac-2D/3D) calculates the hydrodynamic variables at each time step and sends it to Sisyphe. Then it uses the asynchronous solution, meaning it considers the bed fixed at the moment when the hydrodynamic variables were computed, solves the sediment equation and give an update of the evolved bed to the flow module. The schematics of the procedure can be seen below (*1. Figure*) (*Tassi, 2017*)
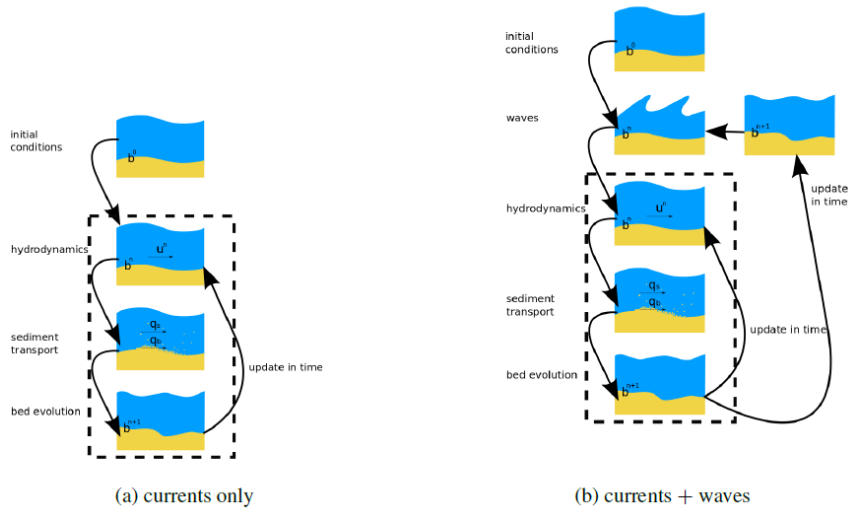
1. Figure: Schematics of the asynchronous solution used by Sisyphe

For *bedload*, we can choose from 10 sediment transport models:

- Meyer-Peter
- Einstein-Brown
- Engelund-Hansen + Chollet-Cunge
- Engelund-Hansen
- Bijker
- Soulsby-Van Rijn
- Hunziker
- Van Rijn
- Bailard

- Dibajnia-Watanabe,

or the users can define their own formula in the subroutines. Some of the formulas above are applicable for bed load only, while others are for total load. Telemac-3D computes the shear velocity, according to the equation below (if the Nikuradse-law is applied), assuming logarithmic profile near to the bed:

$$U^* = \frac{\kappa U_{plane1}}{\ln(33.0\Delta z / k_s)}$$

where U^* *(m/s)* is the shear velocity, κ is the Kármán-constant, U_{plane1} *(m/s)* is the velocity at the first horizontal plane above the bottom, Δz *(m)* is the distance between this plane and the bottom and k_s *(m)* is the Nikuradse friction coefficient. The bed shear stress is calculated with this shear velocity, in order to account for deviations:

$$\tau_b = \rho \cdot U^{*2}$$

After receiving the results from Telemac-3D, Sisyphe proceeds to solve the *Exner equation:*

$$(1-\lambda)\frac{\partial z_b}{\partial t} + \nabla \cdot \mathbf{Q}_b = 0$$

where λ is the bed porosity, z_b is the bottom elevation *(m)*, t is the time *(s)* and $\mathbf{Q}_b$ is the bedload transport vector per unit width *(m²/s)*. $\mathbf{Q}_b$ is composed of Q_x and Q_y and:

$$\mathbf{Q}_b = (Q_{b_x}, Q_{b_y}) = (Q_b \cos\alpha, Q_b \sin\alpha)$$

where the value Q_b is calculated according to the sediment transport capacity and α is the angle between the downstream direction and the sediment transport vector. This deviation depends mainly on the presence of secondary flows and the bed slope.

Based on which transport formula we choose, the current-induced sediment transport rate (φ_b) is computed accordingly. These formulas are functions of the Shields number (Θ):

$$\theta = \frac{\mu \tau_b}{(\rho_s - \rho)gd},$$

where μ is the correction factor for skin friction, τ_b is the bottom shear stress, ρ_s is the sediment density, ρ is the water density, g is the gravity acceleration and d is the grain diameter (d_{50}).

Finally, Sisyphe considers three aspects that can modify the bedload magnitude and direction if the responding key-words are used the steering file. These are the effects of *local bed slope*, *secondary flows* and *the skin friction and drag force components of the bed-shear stress*.

2. Verification

In order to get familiar with Telemac-3d and Sisyphe, I chose to reproduce the well-documented Yen-case study (*Yen et al., 1995)* and compare my results. The laboratory experiment was carried out in a 180° curved channel, with unsteady flow conditions, giving it a complexity and making it a good case for practice.

The test channel was 1 m wide, with a bed slope of 2 ‰ from upstream to downstream. A 11.5 m long, straight upstream part was followed by a 180° bend with the radius of r = 4.0 m, and a 11.5 m long, straight downstream part. The average sediment diameter was d = 0.001m. The initial condition was h_0 = 0.0544 m water depth and steady-flow of Q = 0.02 m^3/s. For the unsteady flow conditions, I chose the "Run 4" from the original experiment (see *2. Figure, 1. Table*).
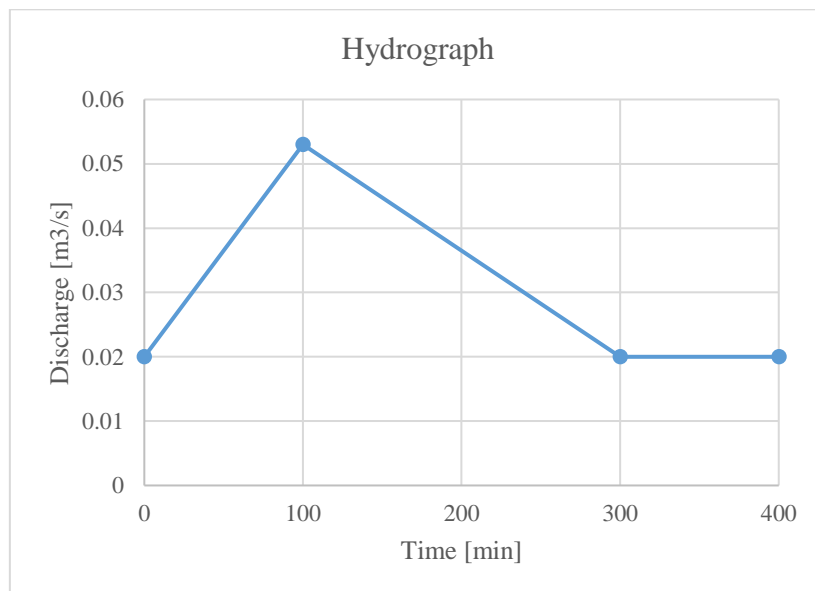
2. Figure Hydrograph of Run 4

Flow [m3/s]	Water depth [m]
0.02	0.054
0.053	0.103

*1. Table Characteristics of the hydrograph*

Since in an earlier study (*Riesterer et al., 2013*) they already used the Yen case to compare the results of Telemac, and since my goal here was just to get control of the settings and see if I can get similar results to the experiment, I created a uniformly rough mesh for the numerical model, with $\Delta x= 0.5$ m, $\Delta y= 0.2$ m cell sizes (*3. Figure*).
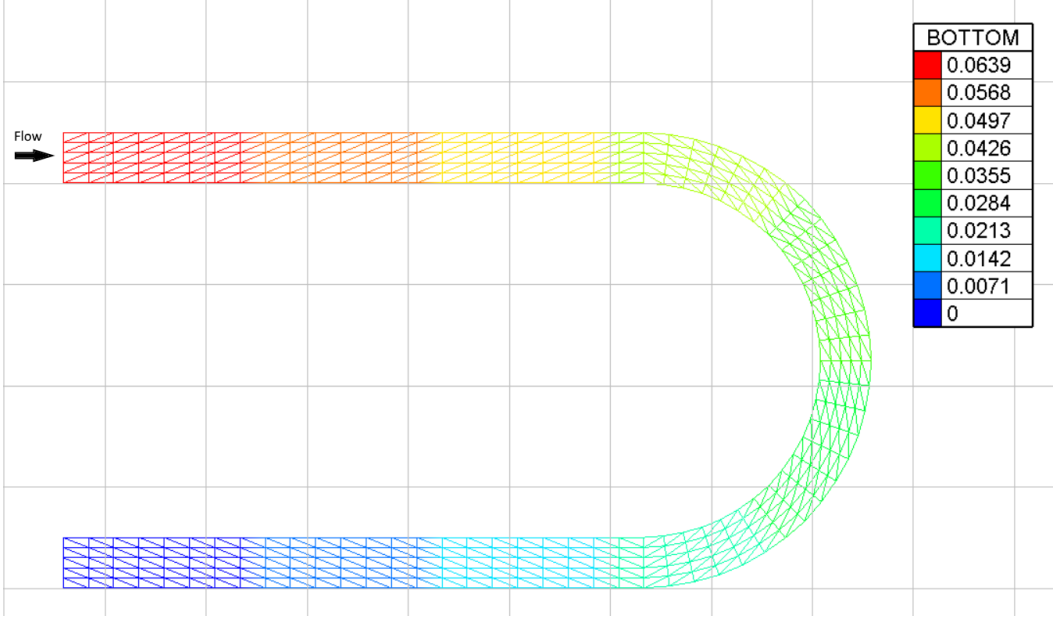

3. Figure The 2D computational grid

Vertically, I defined 10 horizontal levels and applied the normal σ-transformation. The upstream boundary condition was a discharge-time series (*2. Figure*), while on the downstream a Q-H curve was given. The initial conditions and sediment parameters also followed those of the experiment. The non-hydrostatic version of Telemac was used. I applied the Smagorinski model for horizontal and mixing length model for vertical turbulence. The law of bottom friction was calculated according to Nikuradse, with $k_s = 0.0035$, but for the walls I did not specify one. In Sisyphe, the secondary currents were taken into consideration, as well as the effect of bed slope. For the bed load transport formula, I tried all of them to see the differences.

As a result in my case, I found that the formula of Van Rijn-Soulsby (and the formula of Van Rijn) gave the closest match with the experiment results (*4. Figure*).
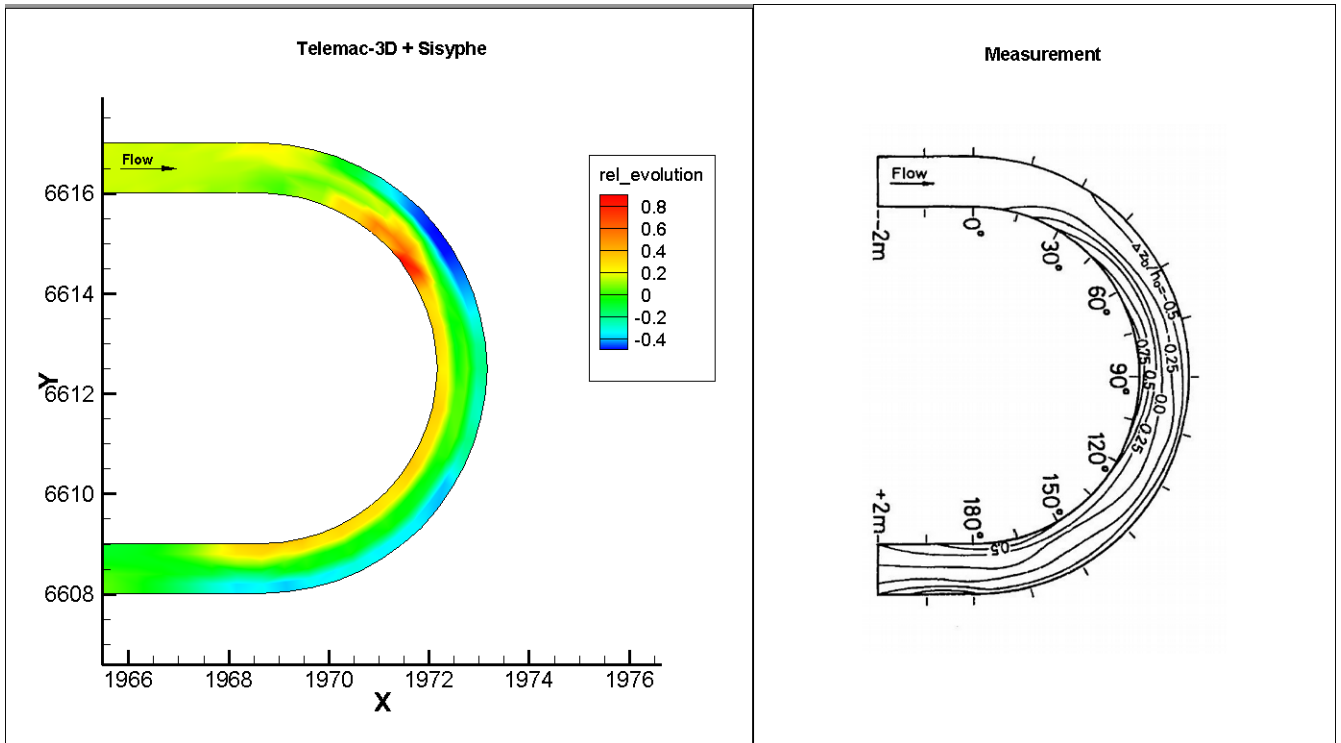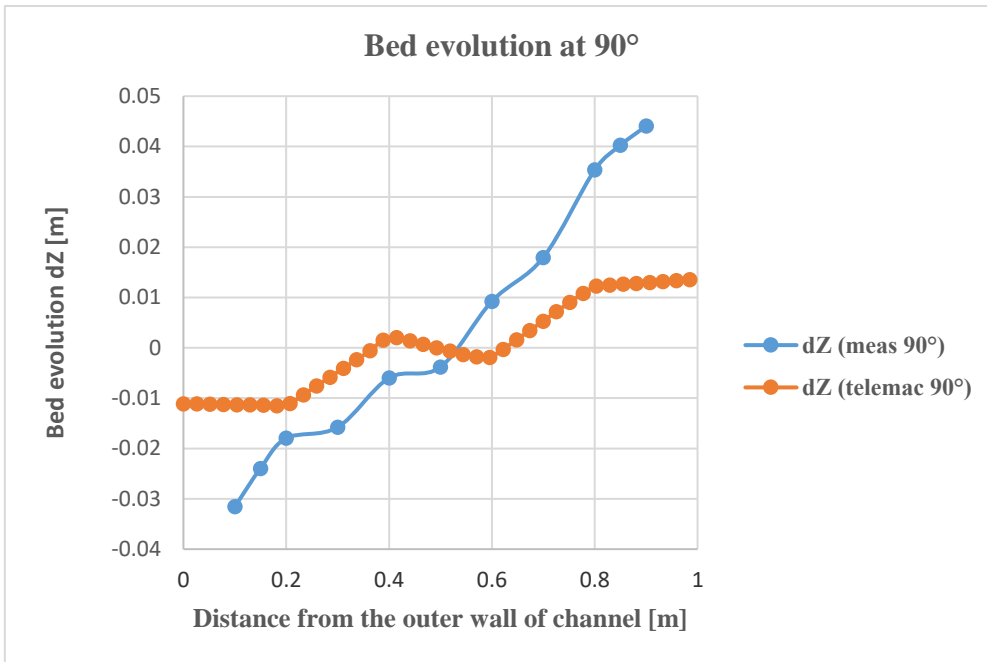
4. Figure Relative bed evolution ($\Delta z_b/h_0$) at the end of the hydrograph (T=400min). Telemac reproduced the deposition in the inner part, and the erosion at the outer part, but with local differences
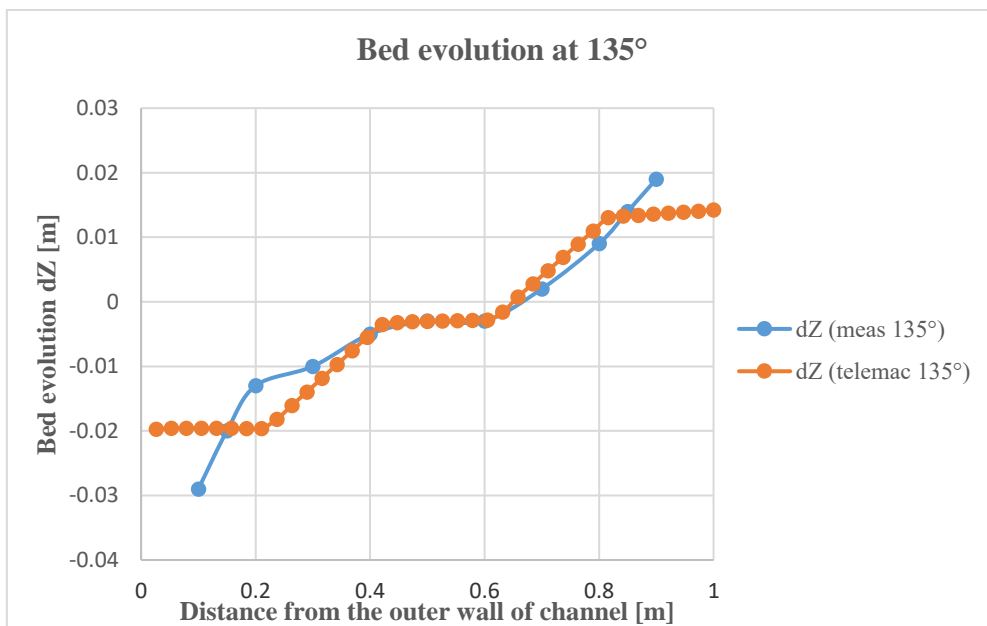
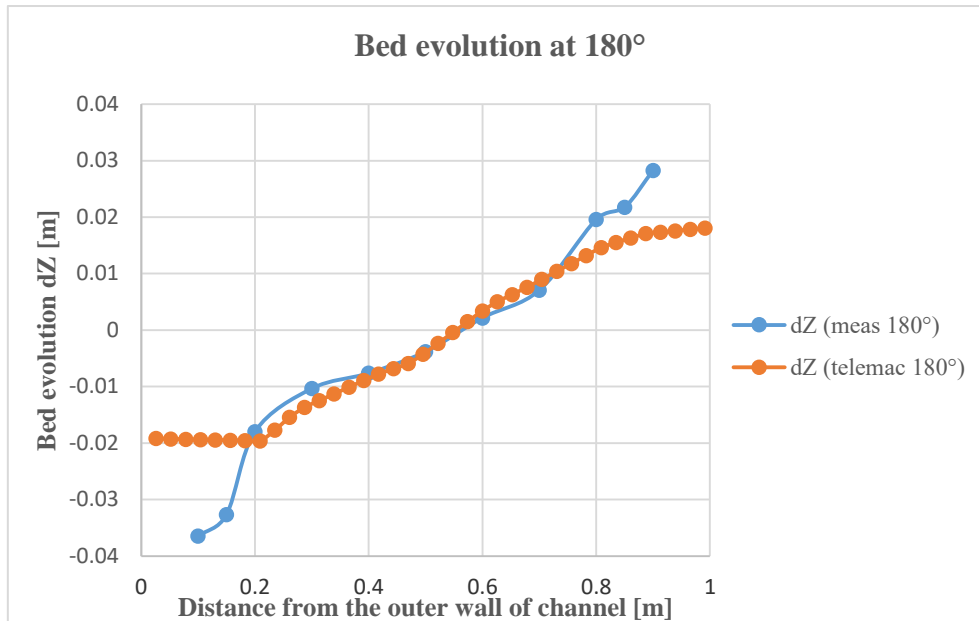The local bed evolutions in the cross of the bend showed the following results:
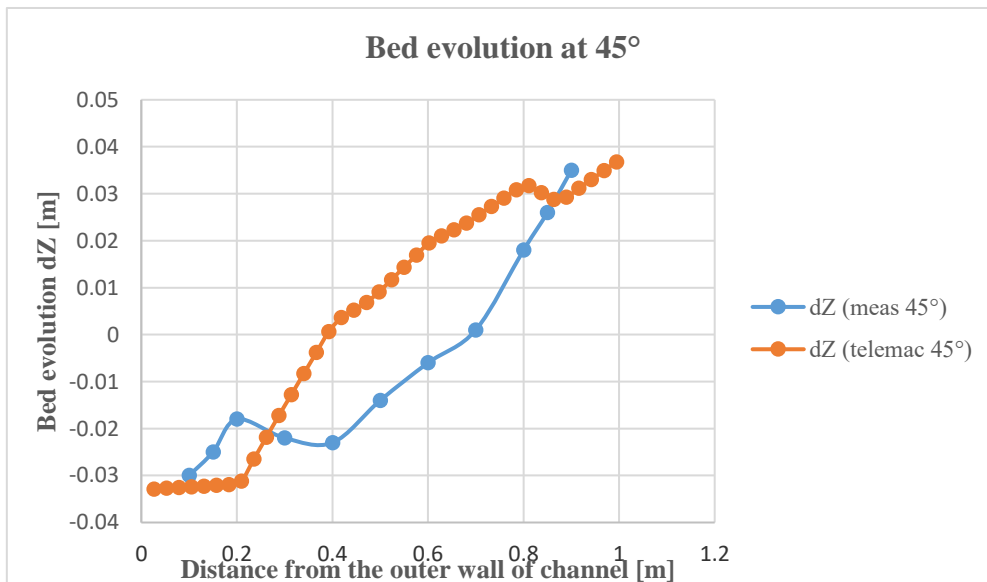
5. Figure Local bed evolution at 45°. Telemac overestimates.

13

6. Figure Local bed evolution at 90°. Telemac underestimates.

7. Figure Local bed evolution at 135°. Telemac shows a close match.

*8. Figure Local bed evolution at 180°. Telemac shows a close match.*

In the mentioned comparison study, they managed to get more accurate results with a refined mesh in the bend, but their overall conclusion was the same. Telemac-3D with Sisyphe managed to give back the typical erosion-deposition pattern and resulted in a good agreement with the measurements (with some local differences, e.g. section 45°).

3. Constructing the model

3.1. The physical model

As previously mentioned, Norway started a project, where the goal is to gain further knowledge on sediment transport processes to improve hydropower plants, affected by high sediment yields. Creating a physical model for simulating flushing scenarios was one of the many steps of the work packages. My task was to try to recreate the model with the help of Telemac-3D and Sisyphe and see how well it is applicable.

After the necessary scaling was done and the test sediment material was chosen, they built the model. The geometry can be seen in the picture below (*9. Figure*). It is a straight, B=0.61 m wide flume with no bed slope. After 9.80 m in flow direction, a sluice gate, with b=0.05 m wide opening, divides the channel into the upstream (reservoir) and the downstream parts (*10. Figure*).
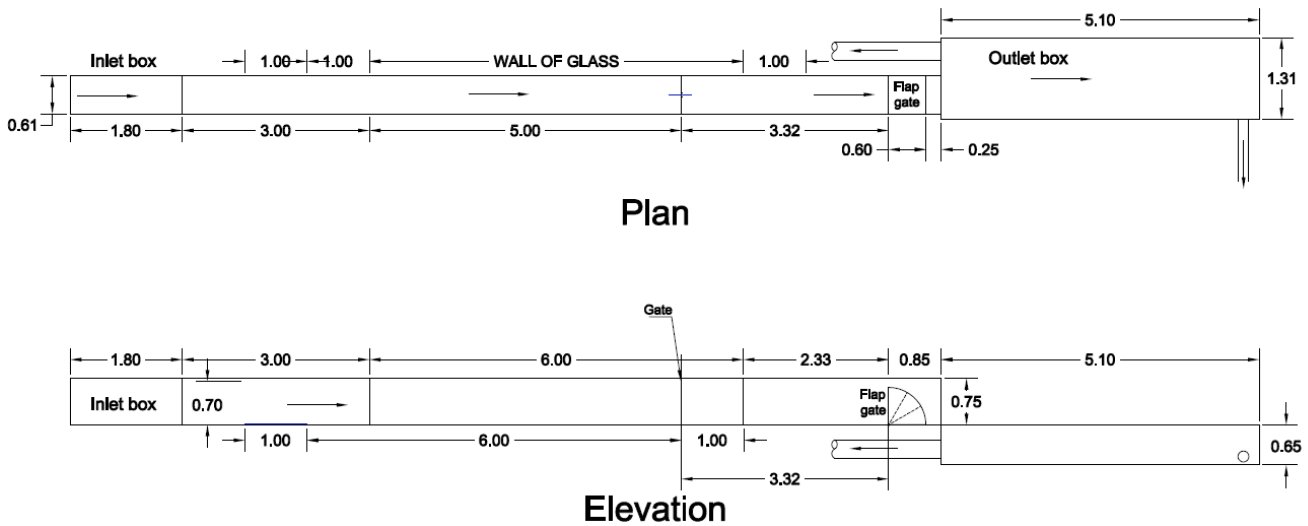
9. Figure Geometry of the physical model. The flap gate was eventually left out. The same geometry was applied for the numerical model as well.
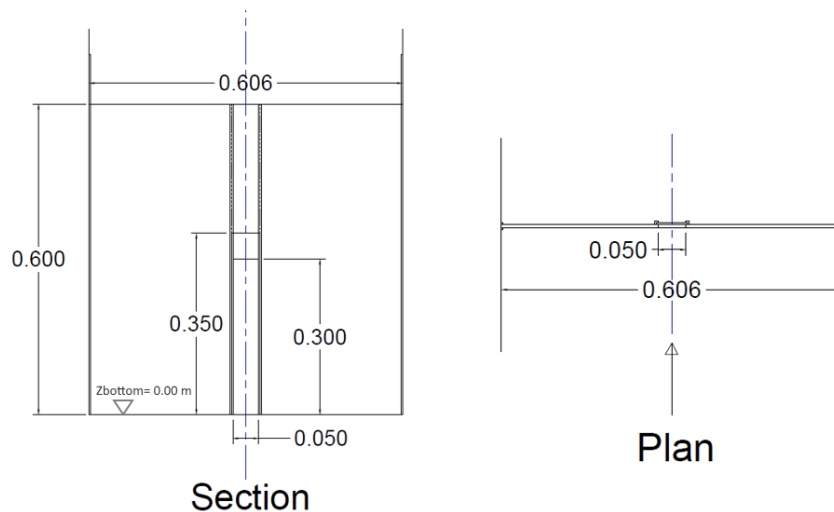

10. Figure The cross gate with the sluice gate, which opens right from the bottom of the flume (z$_b$=0.00m)

The goal was to prove that the chosen lightweighted material can re-enact the behaviour of natural sand sediments settled down above dams and the physical model can be used for further experiments.

The experiment I followed up with the numerical model was a *pressurized flushing* scenario. The goal of this type of procedure is to quickly clean the close area of the sluice gates from the settled sediment. As a result, a so-called *flushing-cone* is formed in the sediment layer around the bottom outlet (*Healey et al., 2015*).

First, the sediment layer with the thickness of t$_s$= 0.1 m (z$_{sedi}$=0.1 m) was placed upstream, in the whole width of the flume, starting from the closed sluice gate until the point we could keep the constant thickness with the amount of available test material (*11. Figure*).
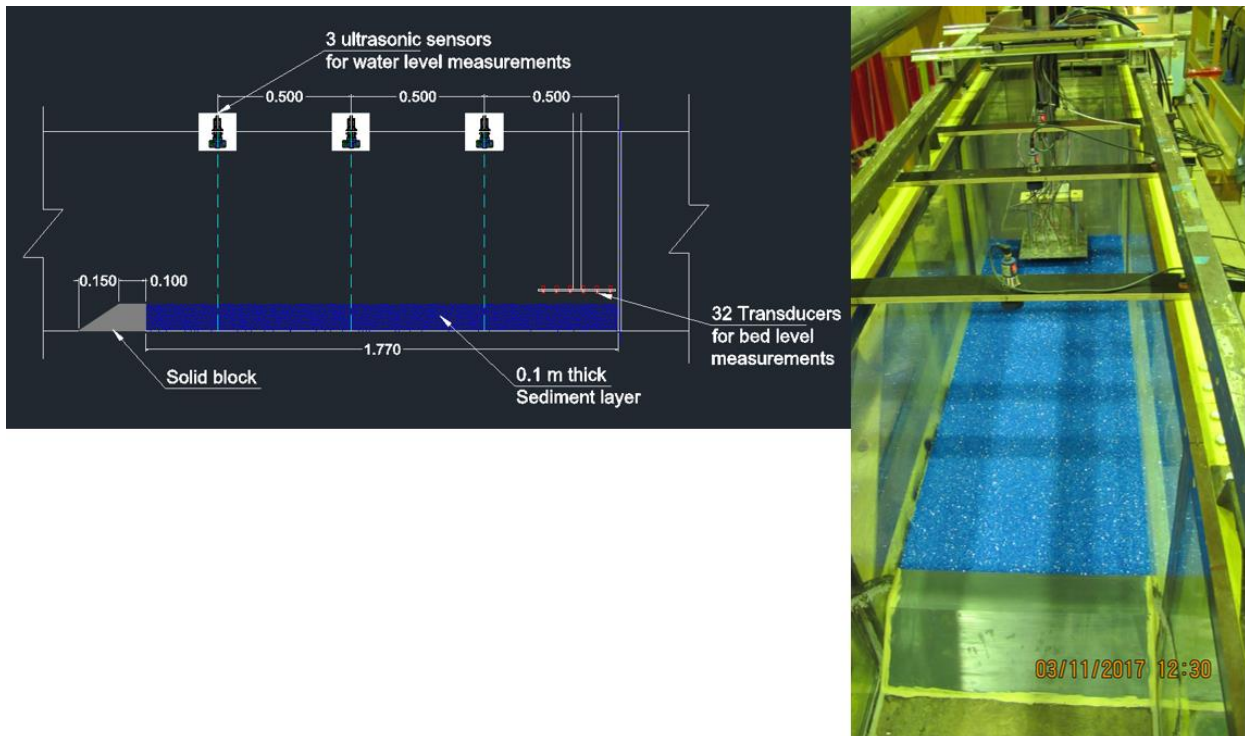
11. Figure The sediment layer (blue) at the start of the experiment. The positioning and type of the measurement equipment can also be seen. The solid block was eventually left out and a closing slope was formed out of the sediment itself.

Then, the reservoir got slowly filled up with water until it reached the initial water depth of $h_0=0.15$ m above the sediment layer (meaning $z_0=0.25$ m). The sluice gate was positioned on the bottom of the channel (*as seen in 10. Figure*). There was no water downstream from the gate. Then we started the experiment by quickly opening the sluice gate to 0.05m (from the bottom).

During the experiment, 3 ultrasonic sensors were measuring and recording the water level (*11. Figure*). For measuring the bed evolution around the opening, a plate with 32 acoustic sensors was placed, right at the cross gate (*11. and 12. Figure*).
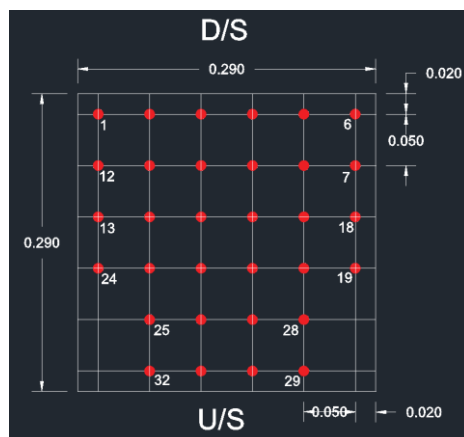
12. Figure Distribution of the acoustic sensors

The duration of the flushing was T=130 s, then the sluice gate was closed. During this period, the water level dropped by 0.0898 m.

The lightweighted, non-cohesive material used in the experiment (**13. Figure**) has a particle density of ρ_p=1180 $\frac{kg}{m^3}$ and dry bulk density of ρ_b=700$\frac{kg}{m^3}$. In its dry state, the friction angle was measured around φ= 30°. The mean diameter is d= 2.4 mm.

13. Figure The applied sediment material.

3.2. The numerical model

Since Telemac is a free surface flow model, my first obstacle was to find a way to model the sudden opening of the sluice gate and the pressurized flow condition of the flushing. Eventually, I decided to use the new, so-called *culvert function* (introduced in April, 2017, v7p2). With this function the users can define *weirs, tubes and culverts* in their domain. My idea was to model the cross gate as a high dike with narrow base, and through it, the sluice gate as a short culvert. As the culvert functionality came out not long ago in Telemac, it has not been used with sediment processes and bed evolution so far. Hence, I needed to see if they can be linked or not. To understand my decisions during the thesis and their results, I would like to shortly introduce how the *culverts* work in Telemac.

Telemac handles culverts as two points (sink/source), between of which the flow can occur. The discharge of this flow depends on the difference between the water levels at each point. Generally, it is calculated according to the formula below:

$$Q = A_0 \left(\frac{\left(2g(upstream\ level - downstream\ level)\right)}{\sum C_i} \right)^{1/2}$$

where:

Q [m³/s]: discharge of the culvert

A_0 [m²]: inflow area of the entrance

g [m/s²]: gravity acceleration

upstream, downstream level [m]: water elevation at the inflow/outflow point

C_i [-]: loss coefficients of the culvert

In regards of considered losses in $\sum C_i$, the equation above slightly varies, based on the type of the occurring flow. Telemac categorizes the flow and accordingly *calculates the discharge at every time step*. There are *6 types of flows* implemented (*14. Figure*) (*Mattic, 2017*).
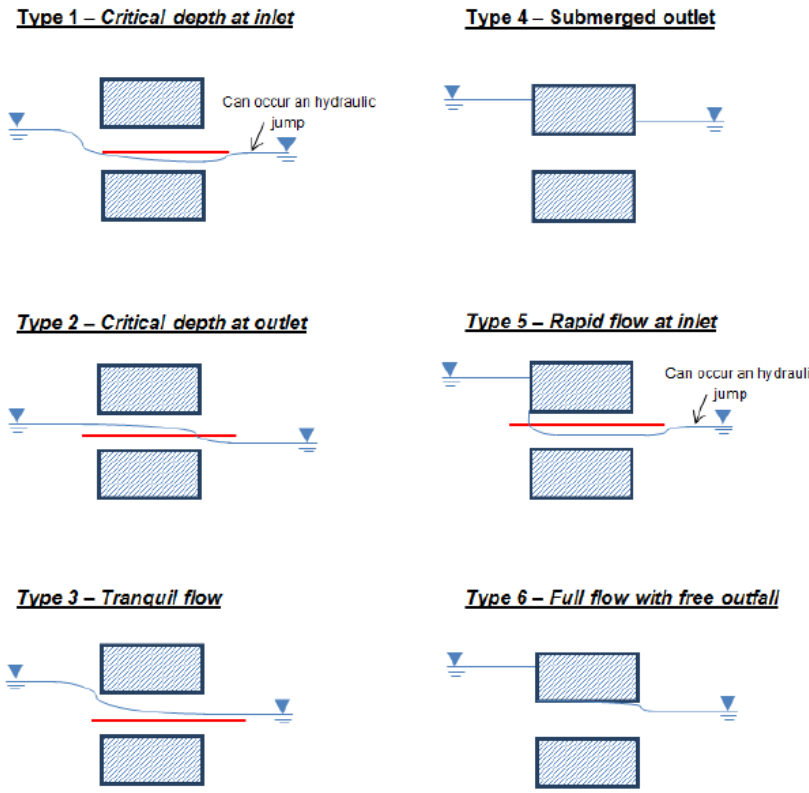
*14. Figure The subroutine checks the responding criterias and categorizes the flow to calculate the discharge.*

The user has to define the following parameters, so that the program can check the criterias:

- *relaxation coefficient* (e.g. R=0.4, means that the discharge at the current n time step is the mix of 40% of the actually calculated and 60% of the previous time step's discharge)
- *node numbers* of the 2D mesh, where the sink (I1) and the source (I2) is wished to be placed
- *direction of the flow (CLP)* through the culvert (options: no flow, only from entry to exit, only from exit to entry, both direction can occur
- *head loss coefficient when the node works as entry (CE1, CE2), head loss coefficient when it works as exit (CS1,CS2)*
- *culvert shape* (round or rectangle) *(CIRC)*
- *length* of culvert *(LONG)*
- *Manning coefficient* for the material of the culvert *(FRIC)*
- *width* of culvert *(LRG)*
- *height* of culvert at the entry *(HAU1)* and height at the exit (*HAU2*)
- *elevation of the culvert entry (Z1) and elevation of the culvert exit (Z2),* meaning these are the elevations of the sink/source points. Telemac places the points on the closest horizontal plane to the given elevations!
- *angle* of culvert with axis x (*d1, d2)*
- *angle of the entry (a1) and exit (a2)* with the bottom
- *linear head loss coefficient (LBUS)*
- *correction coefficients for flow type 5* (because the discharge coefficient is generally lower than in the other cases) *(CV5, C5)*
- *coefficient to differentiate between flow type 5 and 6 (C56)*
- *head loss coefficient due to presence of valve (CV)*
- *head loss due to trash screens (CT)*

Transporting the tracers through the culverts works with the same idea as with the flow (source/sink terms). The *concentration* disappears at the sink, and an equal concentration appears at the source point.

 In the numerical model I followed the geometry of the physical model, so I chose the geometric parameter for the culvert accordingly, with 5cm x 5cm rectangular opening at both sides, starting from the bottom elevation and 2cm length in the flow direction and I defined the sink and source point in the middle of the flume.

The geometry of the complete numerical test flume can be seen in **15. *Figure***. For the cross gate, I elevated the bottom according to the physical model. There is no bed slope, just as in the experimental flume, with the exception where the collecting box starts. As there were no measurements downstream available to create a liquid boundary condition, I created the box ending, where the water flowing through the sluice gate can gather without effecting the water level at the exit of the culvert.
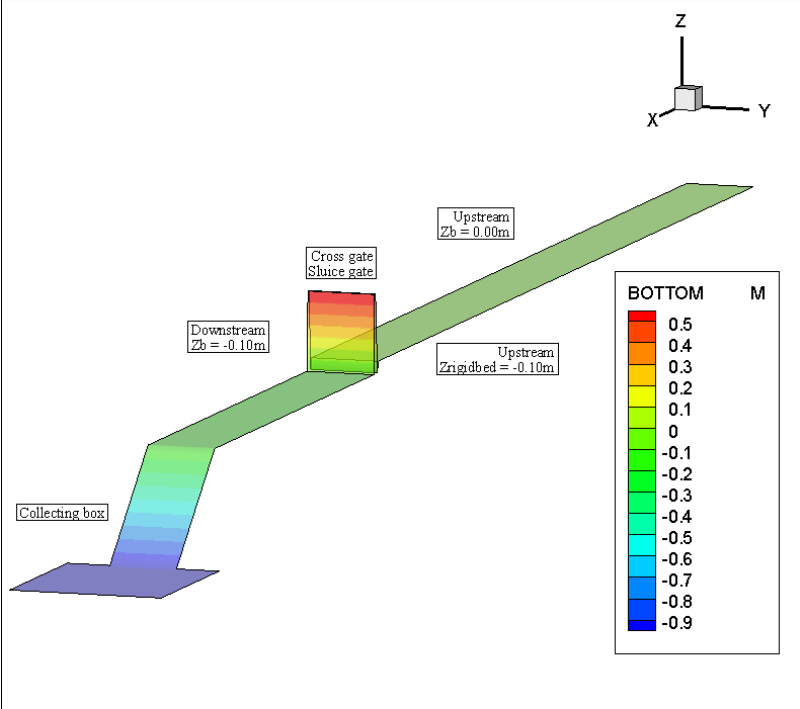

15. Figure The geometry of the numerical model

When defining the bottom geometry, I edited the noerod.f subroutine to define the bottom of the test flume as rigid bed. Starting with the cross gate and through the downstream, the bottom itself was the rigid bed, while I placed it under the bed with 10 cm, on the upstream (to get the initial sediment layer thickness of 0.1m). However, since the culvert functions as a source/sink link, I could not place them under the bottom (the mesh), as in the experiment, where the sediment is covering the opening initially. So I had to create a slope between the cross gate and the sediment layer (**16. *Figure***).
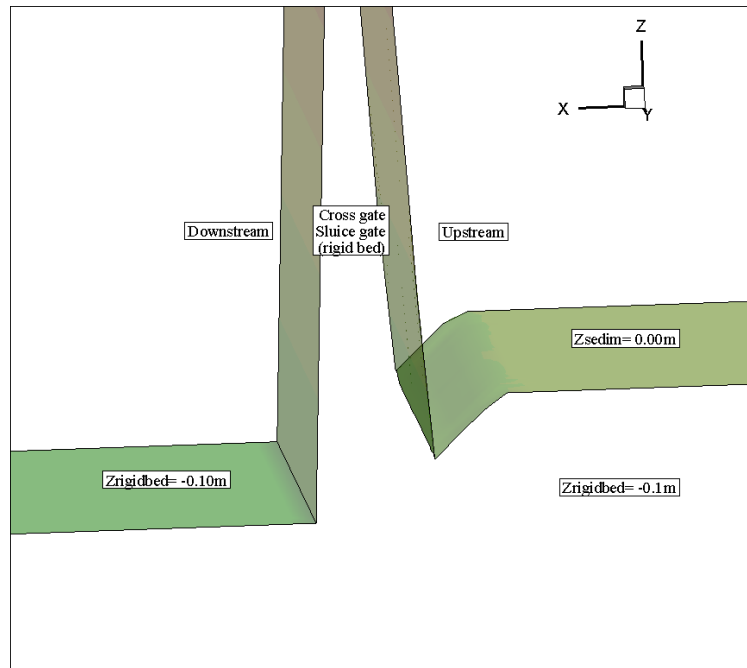
16. Figure Connection of the upstream and the cross gate with sediment slope

With the unstructured 2D mesh, I followed the same method as in the verification phase, I used a rough computational grid, because my goal was to test if the method and Telemac is going to work or not in this situation, and for now I was not concentrating on accurate results. Hence, I defined a mesh size of 2 cm in the area of the cross gate and the sluice gate, and used 5cm further from there (*17. Figure*). I knew that the downstream and the collecting box would take up more computational time and resource, but I needed to see what happens with the flow when it exits the culvert and as I mentioned, I couldn't give a well-established downstream boundary condition at that time.
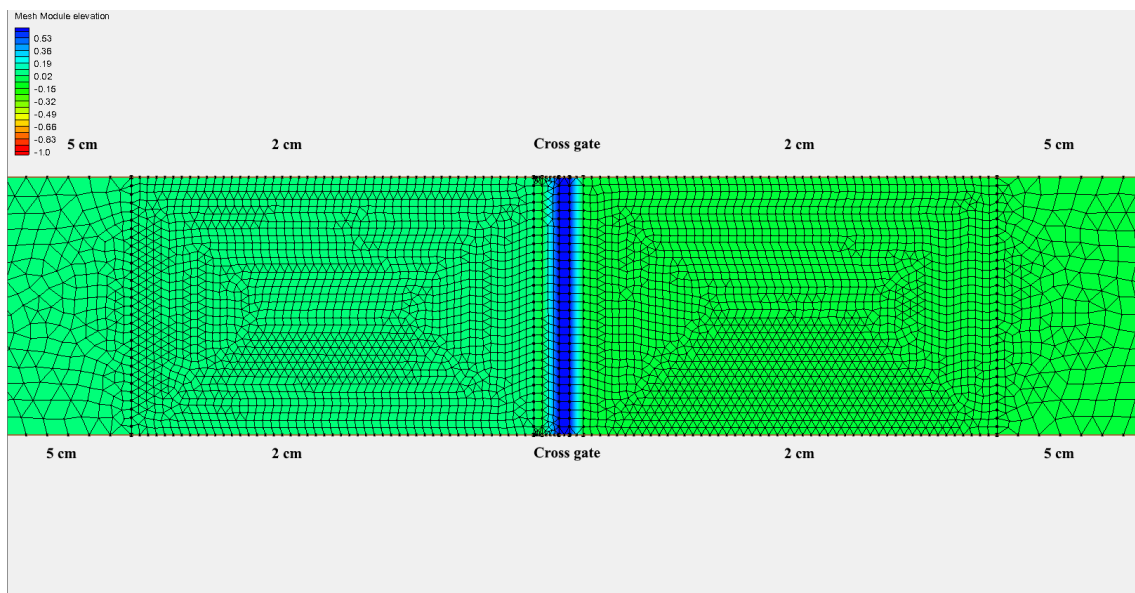

17. Figure 2D mesh distribution zones

22

To create the 3D mesh, I defined 10 horizontal planes with σ-transformation and gave them proportions in the condim.f subroutine. I refined it near to the bed, as it can be seen in the picture below (*18. figure)*. Hence, the horizontal planes are following the change of the water level, but keep the given proportions (as percentage of water depth).
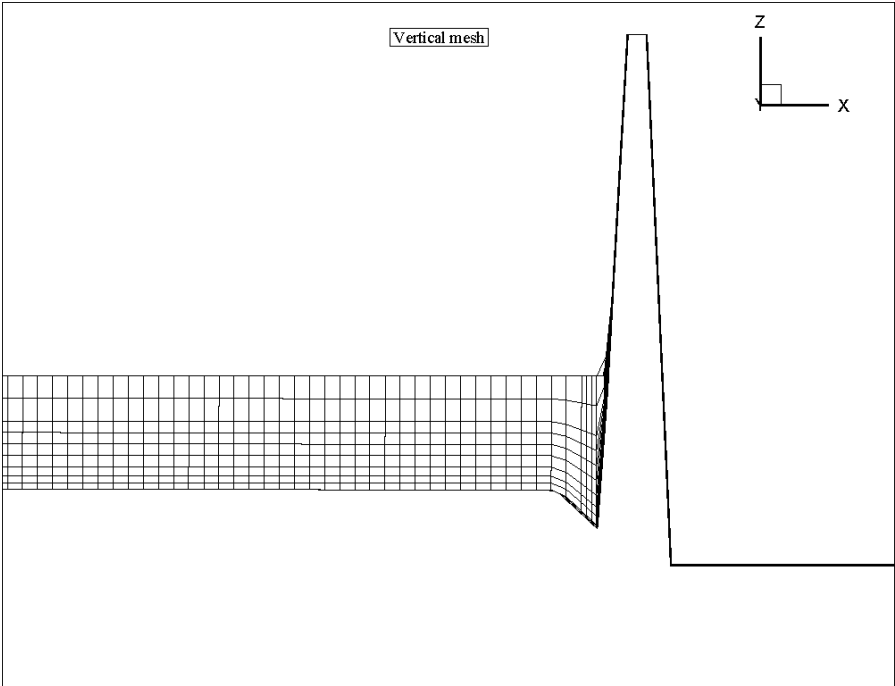
18. Figure The horizontal planes in the beginning of the simulation.

The initial conditions (*19. Figure*) were the same as the experimental ones. The whole domain was surrounded by solid boundaries, no open boundary was given.
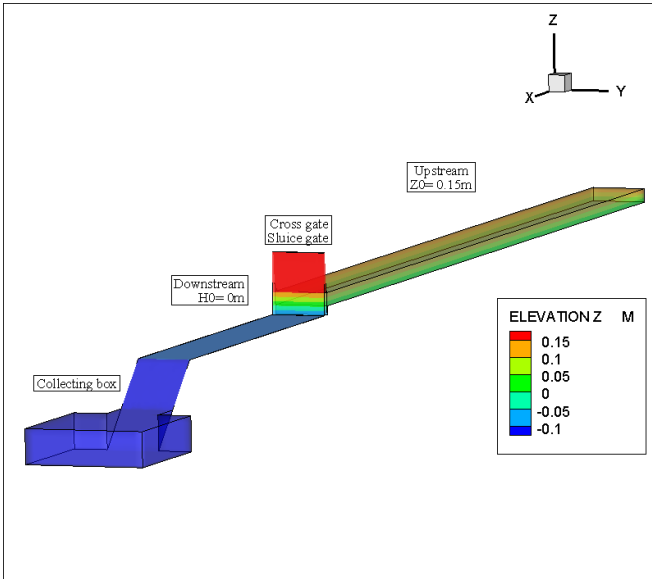
19. Figure Initial conditions: 0.15m water depth upstream and no water downstream.

After establishing the geometry, I proceeded to choose the settings for Telemac-3D. As in the Yen case, I chose the Smagorinski (horizontal) and mixing length (vertical) turbulence models and used the non-hydrostatic version of Telemac. For the bottom friction, I applied the Nikuradse law, with $k_s = 0.0072$ (based on *Van Rijn*, $k_s = 3d_{90}$) and I didn't define wall friction.

In Sisyphe, I considered the slope effect (because of the forming cone) and gave the friction angle of $\phi = 43°$. This choice was based on previous experiment runs in the laboratory, as we wanted to give the steeper wet friction angle and not the dry one (30°), but this should be changed in the future (see later why). Sisyphe takes the friction angle into account for slope stability if the option is used (sediment slide). For taking into account the deviations, the Talmon formula was used, with $\beta = 0.85$. For the bed load transport formula, I chose the Soulsby-Van Rijn method, based on my experience in the validation. For the non-cohesive bed porosity, I calculated and used n=0.407. The simulation period was T=130s as of the experiment, and the t=0.05s time step gave stable runs. The Shields-number was calculated by Sisyphe according to the equation mentioned in *II. 1.2*.

After all of the above mentioned settings, I started to calibrate the model. I had the water depth time series upstream from ultrasonic sensors, and I intended to use them, so I can calibrate the culvert and get the right discharges in time. Out of the 3 sensors, the middle one (in 1 m from the cross gate) proved to be the most reliable, so I also took the simulation results at that point. For the parameters of the calibration, I decided on changing the relaxation and the C5 (correction coefficient for head loss in case of flow type 5). My reasoning was, that I hypothesized the flow would follow the type 5 (14. Figure), where the discharge is calculated as:

$$Q=SECBUS(N)*SQRT(2.D0*GRAV*(S1-RD)/(CORR5*CE1+TRASH))$$

where:

SECBUS [m^2]: area of culvert entrance

GRAV [m/s^2]: gravitational acceleration

S1 [m]: upstream water elevation

RD [m]: average of the culvert entry and exit elevation

CORR5: correction coefficient for head loss

CE1, TRASH: head loss at the entrance, and loss due to trash screens

As it can be seen, in this case only the correction coefficient proved to bear significance and the relaxation coefficient, as I previously mentioned. The calibration resulted in ***20. Figure.***
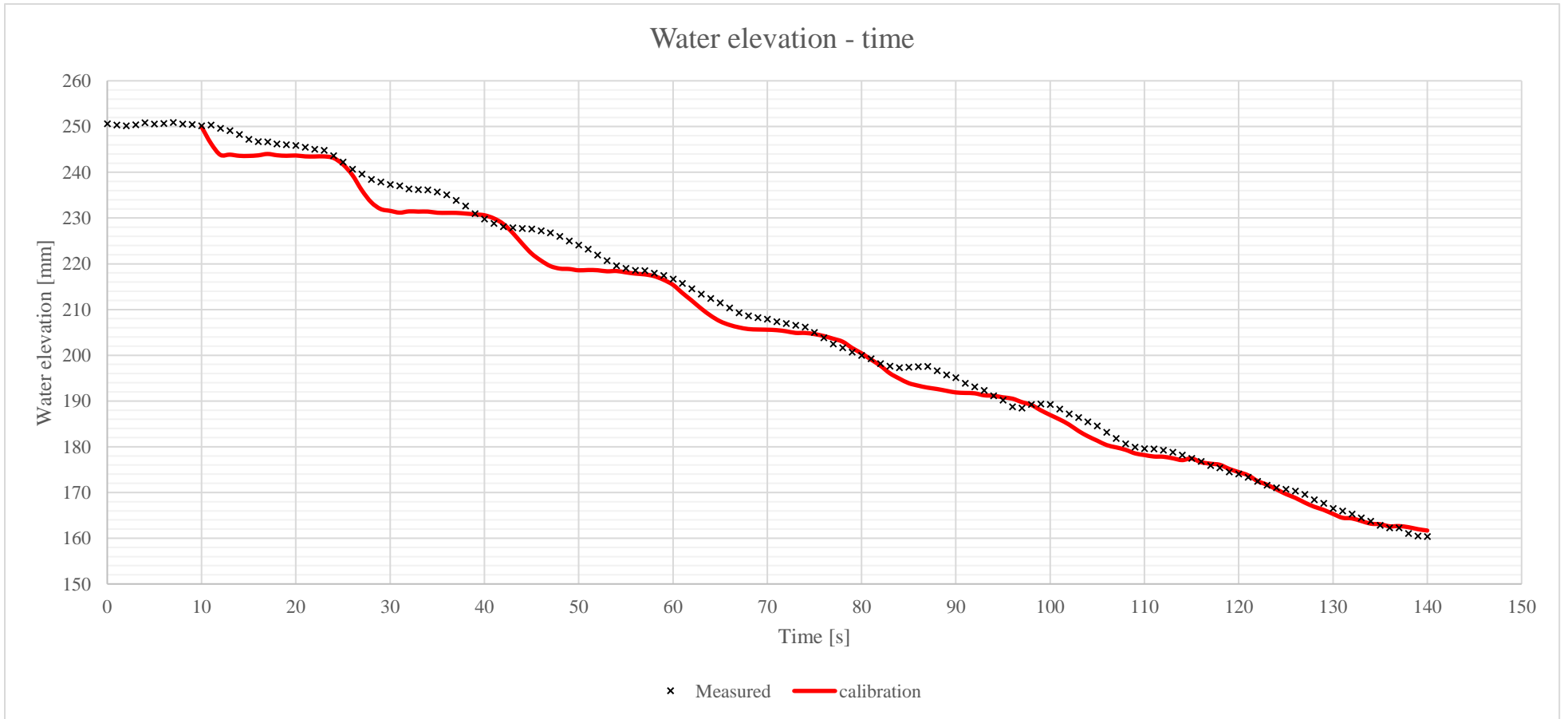
20. Figure Calibration of the culvert, based on the water level changes in time. The sluice gate was opened at T=10s.

The calibration showed good result with the measured data. The simulation even seems to follow the "step-like" decrease. The culvert parameters of the successful calibration can be seen in *21. Figure*.

```
Relaxation, Number of culverts
0.8 1
I1   I2   CE1 CE2  CS1  CS2  LRG  HAUT1 CLP  LBUS   Z1    Z2   CV    C56   CV5  C5   CT   HAUT2  FRIC   LONG    CIRC   d1  d2  a1  a2
3478 1620 0.5 0.5  1.0  1.0  0.05 0.05  1    0.0135 -0.1  -0.1 0.5   1.0   1.0  1.6  0.0  0.05   0.015  0.02   |0      0.0 0.0 0.0 0.0
```

*21. Figure Culvert parameters of the final run.*

As the function is still freshly implemented, beside the two validation cases it has not been widely used. Both cases had tidal rivers and flood plains, where the culverts connected the areas. When I first tried to use the culvert function, the discharge results were really low, almost 0 and they had no relevance to the water levels. After going through the buse.f subroutine (responsible for the culvert calculations) I found that it calculates the discharges with two methods. One is the previously mentioned one, but the other is a function of the chosen time step, and the parallel component of the flow velocity at the culvert entrance. Telemac takes the minimum value out of the two calculated discharges. I suppose it is because it was only used in riverine simulations so far and they didn't want to let through the flow with the incoming velocity of the river. However, in my case it is quite the opposite scenario. Since I have still-water at the entrance, my velocities are close to 0 and as the program took the minimum, the discharges were low. Hence, I needed to change the code in the subroutine. Only after this, was it possible to reach the calibration results.

With the setup described above, the number of elements was 9676, and it took 2 hours and 50 minutes with an i5-2.3 GHz laptop to simulate T=130s.

4. Analysing the results

As the discharges now seemed to be correct, I proceeded to analyse the results the model delivered. First, I observed the velocity vectors, how they act around the culvert (*22. Figure*).
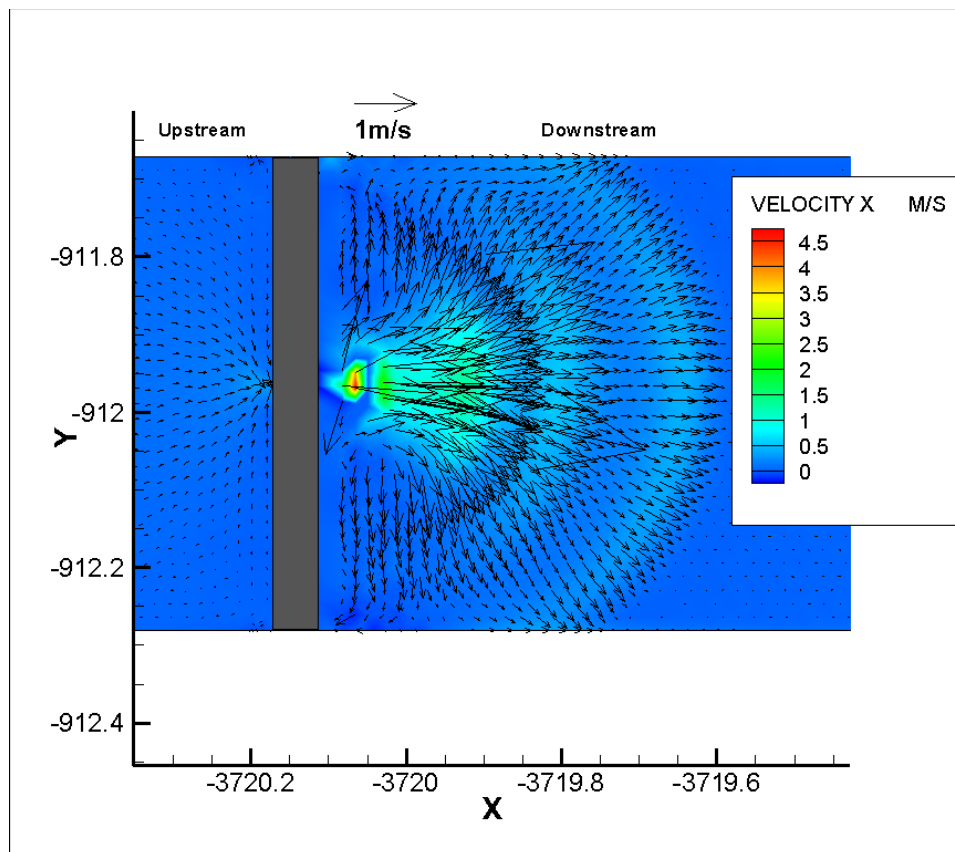
*22. Figure 2D velocity vectors around the culvert (T=1s)*

Since I didn't have velocity measurements, I could not compare these results exactly, so I accepted what the model delivered for now, based on the discharge calibration and the look of the flow vectors.

Next, I was looking at the bed evolution and if the cone was formed or not, but eventually Telemac managed to simulate a cone-formation (*23. Figure*). This meant the theory was working so far.
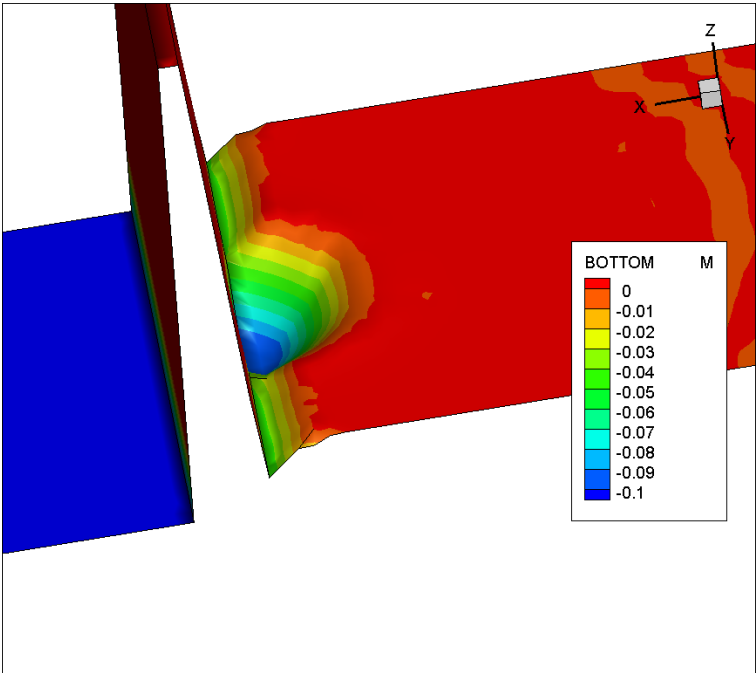

23. Figure The sediment layer with the final cone (T=130s). Along axis Y, the initial slope can be seen.

Reaching this point and seeing the formation of a cone gave reasons to go on with this method. After briefly examining, the velocity vectors at the cone seemed to behave reasonable (*24. Figure*).
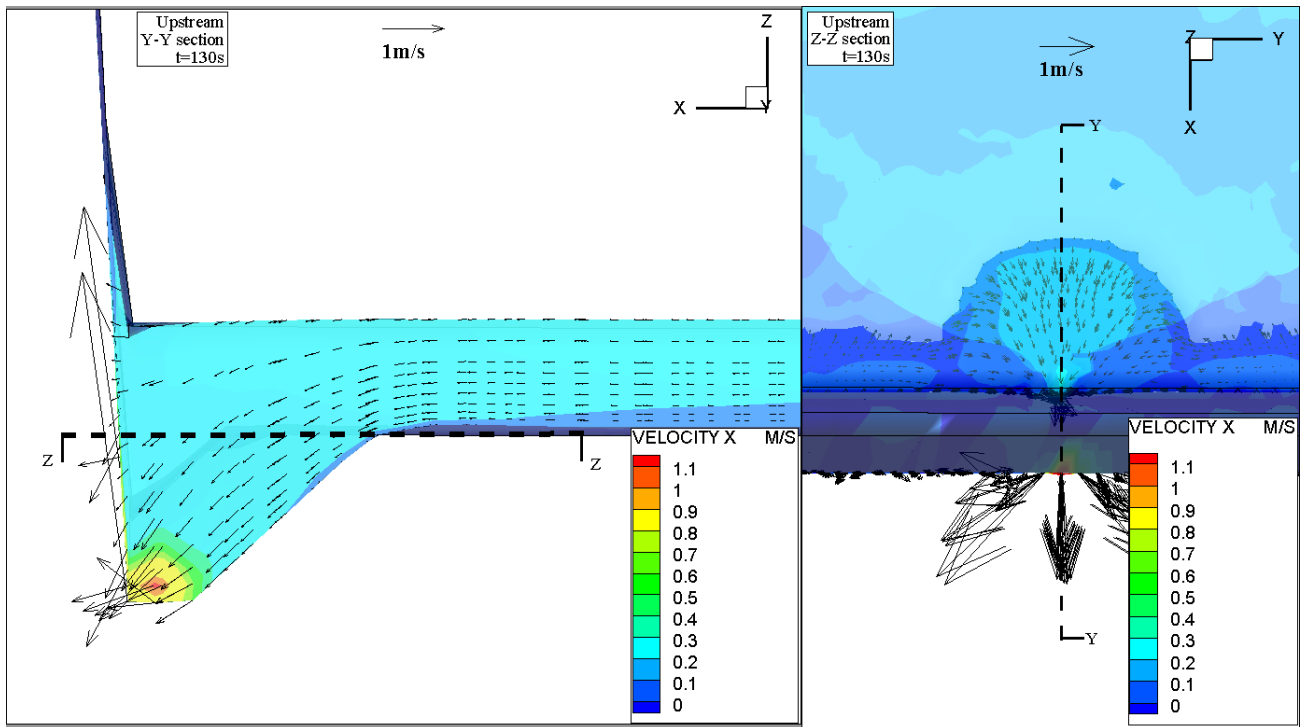
24. Figure Velocities around the cone, pointing towards the culvert.

What I could compare, was the sizes of the cone. However, I had to consider that because of the initial difference in the sediment layer along the cross gate (sediment slope), I should not expect too big similarities. That is why first I compared the sizes parallel to the flume, at the end of the test. (*25. Figure*) R1 is the radius of the top of the cone where the slope starts, while r1 is the radius of the area where the cone touched the bottom rigid bed and all sediment had been washed away. First of all, if we just look at the appearance of the cones, it can be seen that in both simulation and measurement case the cone is not consisted of centric circles, but more of elongated egg shapes. Secondly, the difference between the radiuses is 2-3 cm, which could have been expected, if we look back to the Yen validation case, so I considered it a good match so far. In case of the difference along the cross gate, it would be several more centimetres (5-7).
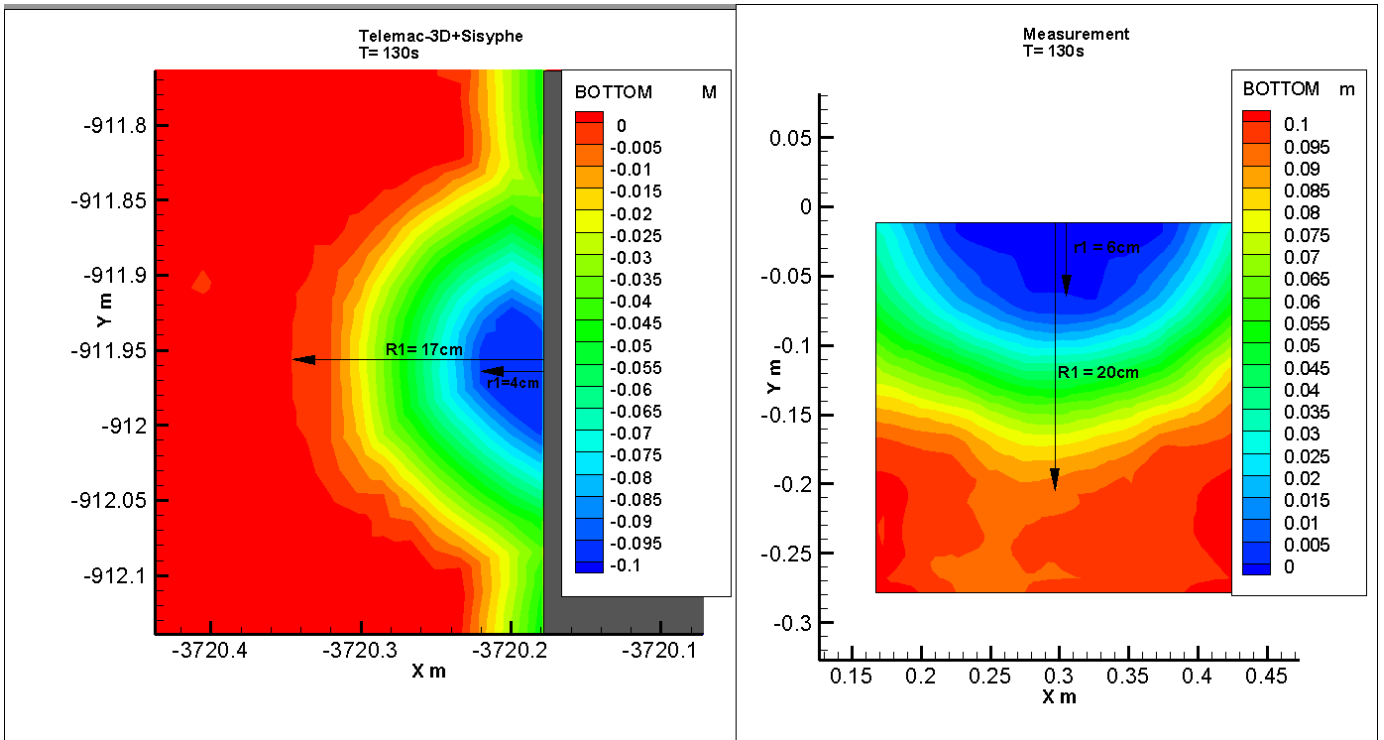
25. Figure Comparison of the cone radiuses parallel with the flume, at the last time step, when the sluice gate was closed.

Additional experience was that how comparing the results can slightly depend on the chosen interpolation method for the measured data. In 25. Figure the measured data was represented with Kriging, while in *26. Figure* the scatter interpolation was used in Matlab.
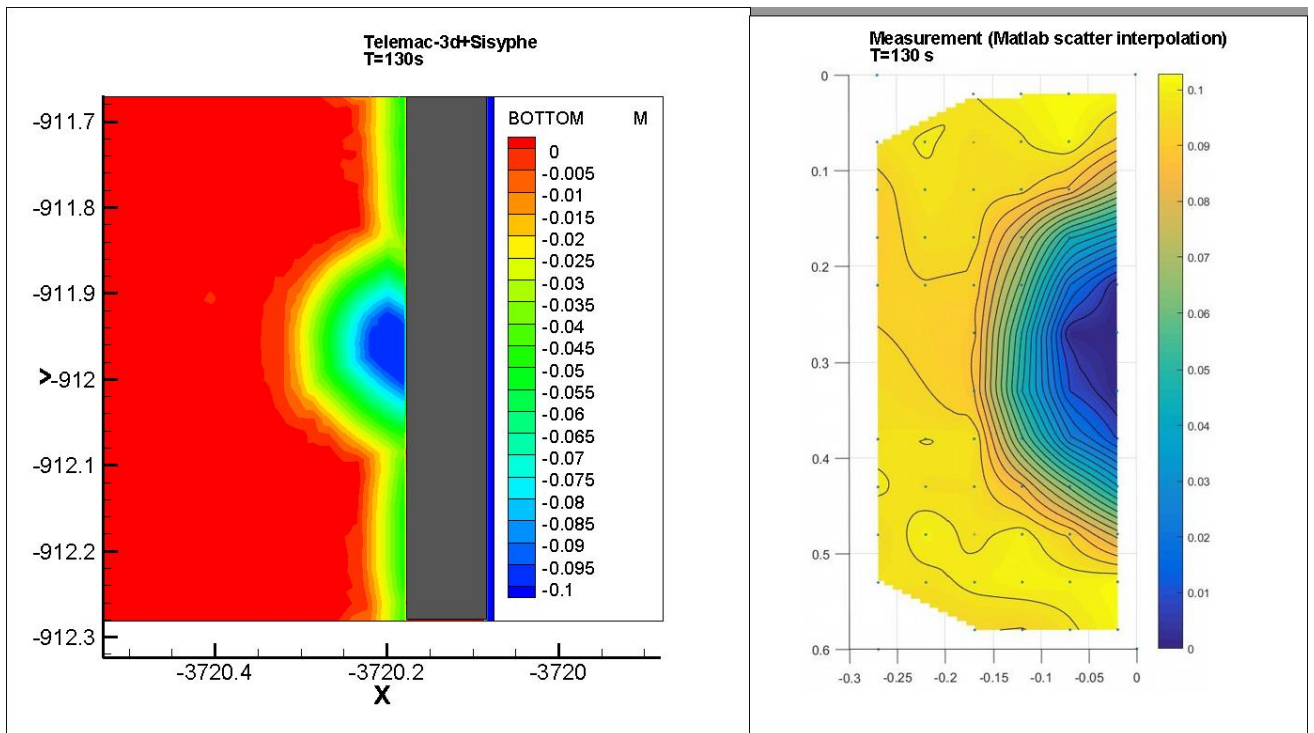
26. Figure Simulation result compared to the measured data, using scatter interpolation

30

Hence, it is worth comparing the results to the photos, taken during/after the experiment (*27. Figure*).
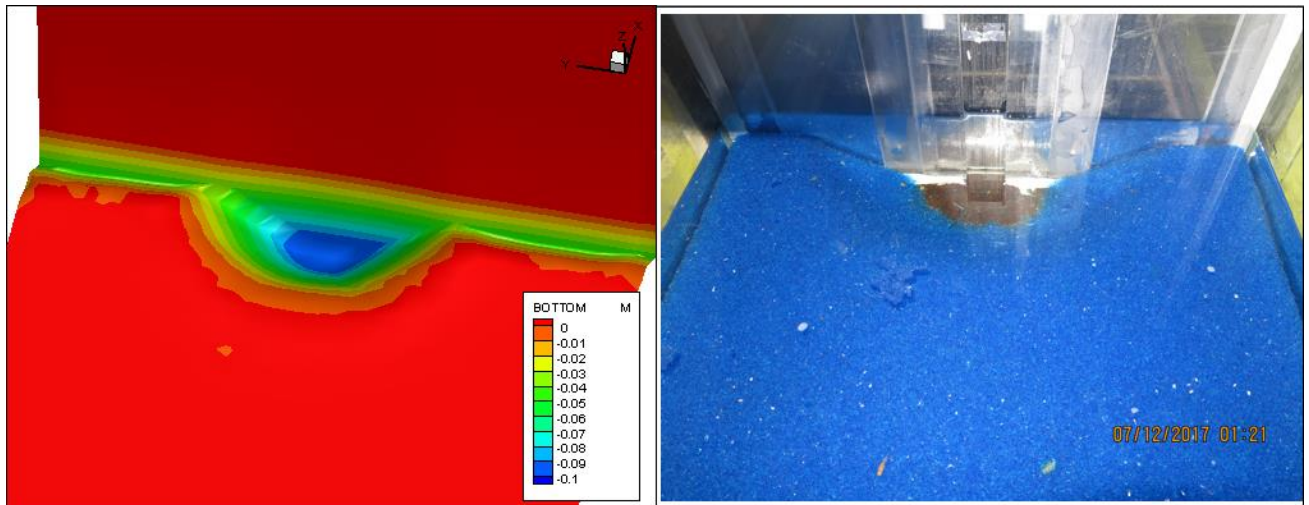
27. Figure The simulation result compared to the experiment result at the end of the run.

Next, I examined the speed of the erosion. For this, I looked for the first moment the cone reached the bottom. During the experiment, it happened after 25 s, while in the simulation it was at 29 s (*26. Figure*). However, the size of the washed out area differs (blanked out with white), I considered this as a good match.
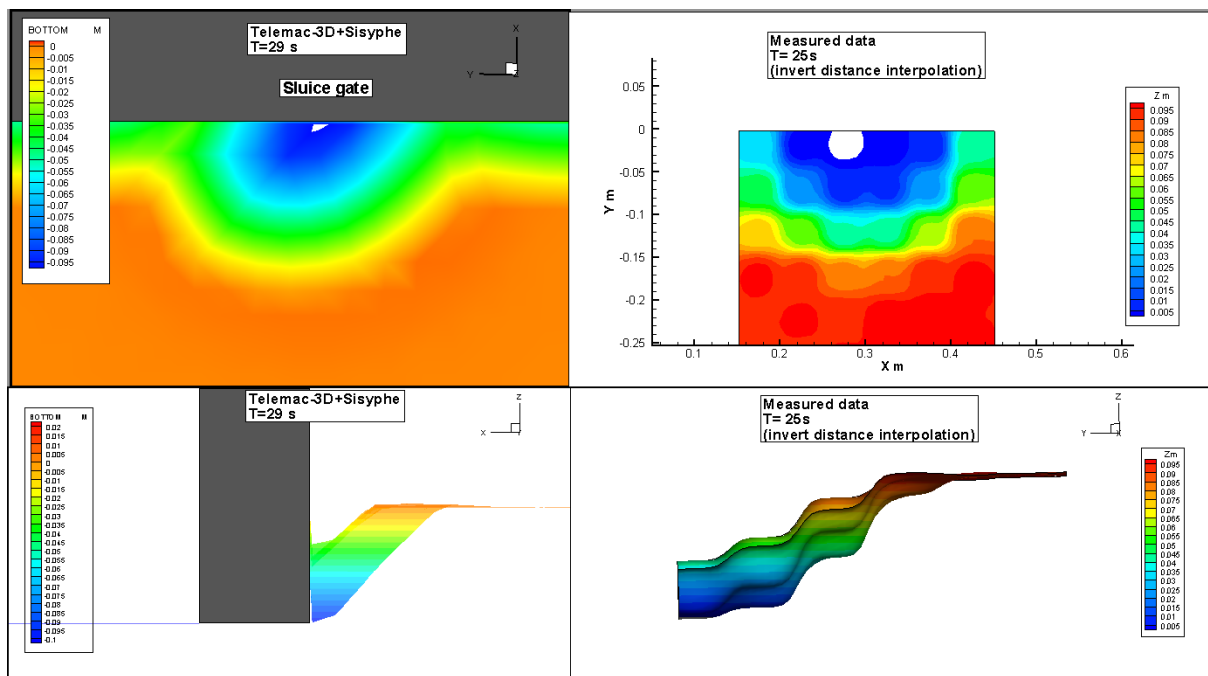
28. Figure The moment when the cone reaches the bottom. Left: simulation, right measurement (with inverse distance interpolation). The area that reaches the rigid bed is blanked out with white.

By this point, it was obvious that the sizes of the cones along the cross gate are not matching with the experiment data and the model needs improvement in this regard.

Lastly, I checked the bed shear stress values around the cone, as they are one of the most important parameters for the sediment transport (*29.-31. Figure*). A pulsation in the bed shear stress values can be observed, as it was in the case of discharges and the water level time series (20. Figure).
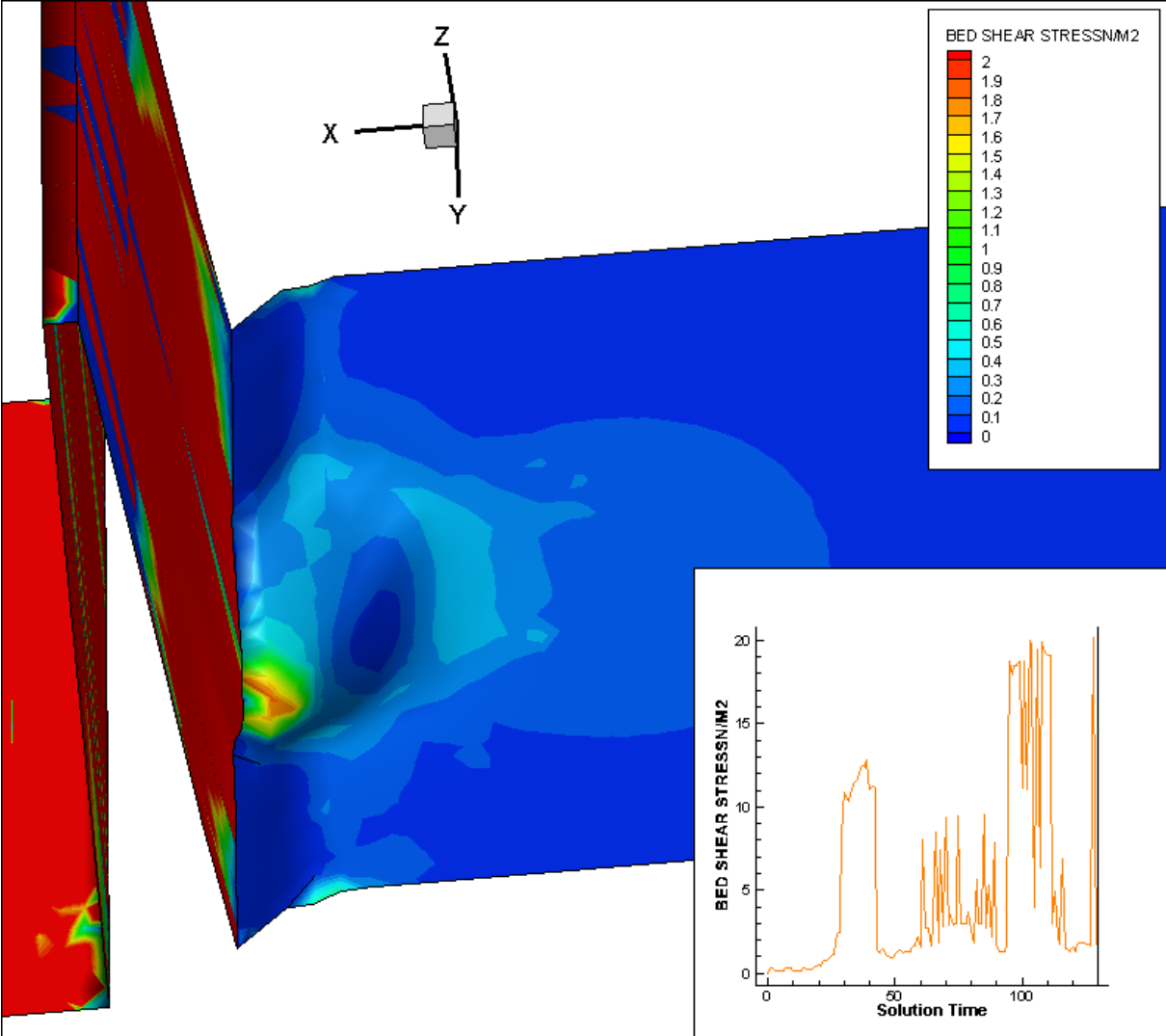
29. Figure The bed shear stress values around the cone at the end of the simulation. In the right corner, the time series of
τ_b at the entrance of the culvert. However, because of the pulsation t=130s seems to be effected, so 30. Figure might be
more accurate (T=128s).
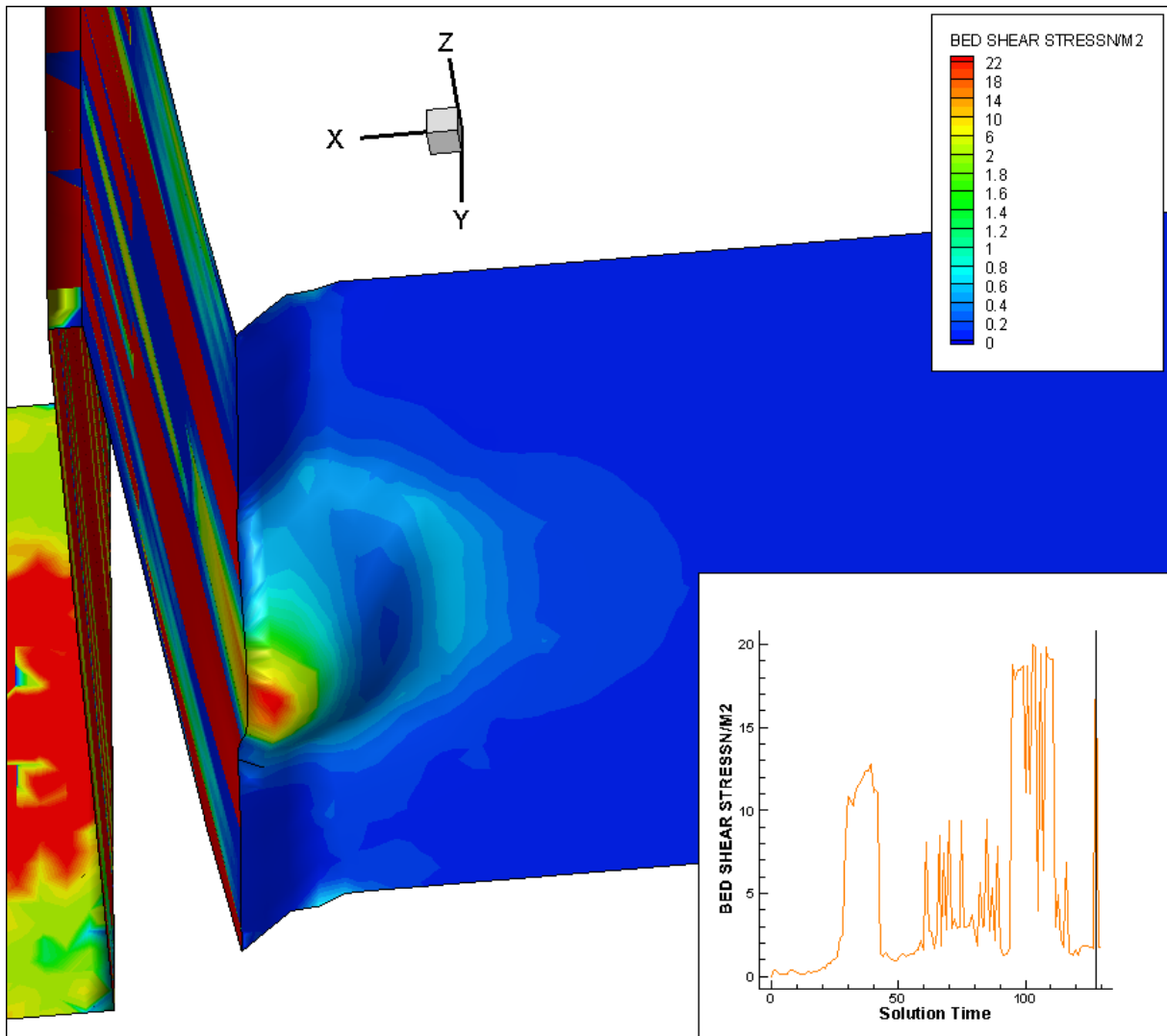
30. Figure Bed shear stresses around the cone at t=128 s (local peak in the time series of the culvert entrance point).

It can be seen that along the edge of the sediment bed slope, I had built in for the previously mentioned reason, the bed shear stresses are somewhat higher than those along the slope, towards the cross gate. From this, I deduce that the magnitude of the cone evolution in this direction is not matching due to the initial sediment slope, which causes a hiding effect for the rest of the slope particles.

5. Evaluation of the model and future directions

The original question was if the free surface flow model Telemac-3D, coupled with Sisyphe can be applied for a sediment flushing scenario and if yes, then what are the limits of doing so. As it was shown, with the newly introduced culvert functionality, the sluice gate discharges of the pressurized flow can be well simulated and a flushing cone can be reached. The sizes of the sediment formation in the flume direction showed a good match with the measurements, as well as the speed of erosion, however they showed different picture along the orthogonal axis.

In my opinion, further improvements can be done to solve the disagreement with the laboratory results and we did not reach the limit of the application, yet.

First of all, a way should be found to avoid creating an initial slope in the sediment layer close to the cross gate, because (as expected) it may cause a hiding effect and spare the lower part from erosion. Or if that cannot be done, then carry out an experiment with creating a similar slope in the physical model and compare the results after that.

I did not payed too much attention for the turbulence, because the sensitivity analysis was not part of the tasks for now. But as we know, turbulence has a significant effect on the movement of particles, so using the k-ε model could also lead to better agreements.

As I mentioned, I defined the friction angle with the slope effect, based on an approximation and by doing so I limited the slope-stability to that angle. However, the slope angle and the radius of the cone correlates. Keeping this in mind, I would like to rethink the parameter for the (wet) angle of repose.

In the numerical model, the cross gate is actually a part of the bottom. However, I uniformly defined the bottom friction for the domain, giving k_s= 0.0072 friction parameter, which here acts like a wall friction, influencing the flow velocity along the cross gate, while in the physical model that part is as smooth as the other walls of the flume (the ones of which I didn't give friction). The solution for this would be really simple, just redefining the friction zones.

I mentioned before that the culvert is practically a sink and a source point and not a 3D object. This fact might lead to differences from the experiment results in itself, but considering that Telemac places these point on the closest horizontal plane to the defined culvert elevations,

causes them to constantly change their altitude. However, this problem can be eased by defining more horizontal levels, it should be kept in mind while checking the results.

Carrying out velocity measurements could also help enhancing the numerical model.

During the tests, I used a rough mesh distribution. Refining the computational grid may also improve the quality of the results.

All of the above mentioned thoughts lead me to say that I think we reached a better result than one may have first expected, considering the *given* (how the culvert function works and modelling the sluice gate with source/sink points) and *handmade* (initial sediment slope) simplifications and changes. At its current state, the model is able to reproduce the for-and-aft evolution and the evolution speed of the cone, as well as the discharge-time curve of the sluice gate and the pressurized flow, but failed to reach close agreement with the evolution in the other direction. However, I think with the mentioned ideas, it can be further improved for that purpose.

III.   Summary

As we saw, sediment can be a major factor in decreasing the lifespan, efficiency and profit of a hydropower plant. Countries and companies have to have countermeasures against the sedimentation, where high yields are expected. However, this is only possible with a firm knowledge base of these processes. For this, beside field measurements, physical and numerical models are great assets.

In my thesis, I was supposed to test the TELEMAC-MASCARET modelling system for reconstructing a pressurized sediment flushing scenario, which was carried out in a laboratory physical model.

First, I carried out a test run with Telemac-3D (flow module) and Sisyphe (sediment transport module) with the well documented and widely known Yen study, to get familiar with the system. After that, I moved on to the objective.

As Telemac is a free surface flow model, the question was how to use it for the pressurized flushing. Finally, the new culvert function appeared to be a possibility for that. However, both the initial setup and the source code required some changes to make it work. The model managed to show a similar culvert discharge-time curve as the laboratory measurements. The changes in the setup of course led to differences between the laboratory and the simulated evolution results.

In spite of this, the flow conditions and the fact of an evolving flushing cone made the method look promising. The for-and-aft evolution size and the evolution speed showed agreement with the measured data. However, the evolution parallel to the cross gate showed bigger differences.

Beside the evolution, I also represented the velocity and the bottom shear stress field results of the simulation, but without measured velocities, I could not compare them.

Finally, I evaluated the model and presented further steps and ideas to improve it and reach a higher quality agreement between the simulation and the laboratory measurements.

References

Alliau, D., Nunes, P., Peteuil, C., Huybrechts, N., (2016). *Using an eco-friendly flushing event to calibrate 3D sediment transport model through a reservoir: the case study of Champagneux run-of-river dam on Rhone River, France. Proceedings of the XXIIIrd Telemac-Mascaret User Conference. 49-54.*

Eltvik, M. (2013). *Sediment erosion in Francis turbines. Norwegian University of Science and Technology, Faculty of Engineering Science and Technology.*

Guerrero, M., and Rüther N. (2014). *Sustainable design and operation of hydro power plants exposed to high sediment yield (SediPass). Energix 2015-2018. project paper.*

Healey, K. M., Cox, A. L., Hanes, D. M., Chambers, L. G. (2015). *State of the practice of sediment management in reservoirs: Minimizing sedimentation and removing deposits.*

Kondolf, G. M., Gao, Y., Annandale, G. W., Morris, G. L., Jiang, E., Zhang, J., Cao, Y., Carling, P., Fu, K., … (2014). *Sustainable sediment management in reservoirs and regulated rivers: Experience from five continents. Earth's Future. 2:5, 256-280. doi: 10.1002/2013EF000184*

Kopmann, R., Walter, L., Frings, R. (2016). *Cross-sectional variation of bed-load transport. Comparison of measurements and simulations. Proceedings of the XXIIIrd Telemac-Mascaret User Conference. 69-74.*

Mattic, O. (2017). *Telemac3d user Manual.*

Morris, G. L. and Fan, J. (1998). *Reservoir sedimentation handbook: Design and management of dams, reservoirs and watersheds of sustainable use.*

Neopane, H. P. (2010). *Sediment erosion in hydro turbines. Norwegian University of Science and Technology, Faculty of Engineering Science and Technology.*

Olsen, N. R. B., (1999). *Computational Fluid Dynamics in Hydraulic and Sedimentation Engineering. Class notes.*

Olsen, N. R. B., (2014). *A three-dimensional numerical model for simulation of sediment movements in water intakes with multiblock option. User's manual.*

Riesterer, J., Grafmüller, T., Wenka, T. (2013). *Comparison of hydraulic and morphological predictions of Telemac-3D/Sisyphe and SSIIM in curved laboratory channels.*

Robertson, M. J., Scruton, D. A., Gregory, R. S., and Clarke, K. D. (2006). *Effect of suspended sediment on freshwater fish and fish habitat – Background. Canadian Technical Report of Fisheries and Aquatic Sciences. 2644: v + 37 pp*

Santoro, P., Fossati, M., Tassi, P., Huybrechts, N., Van Bang, D. P., Piedra-Cueva, I., (2016) *2D and 3D numerical study of the Montevideo Bay hydrodynamics and fine sediment dynamics. Proceedings of the XXIIIrd Telemac-Mascaret User Conference. 177-188.*

Scheuerlein, H. (1991). *Sediment sluicing in mountain reservoirs. Obernach Hydraulics Laboratory.*

Shen, H. W. (1999). *Flushing sediment through reservoirs. Journal of Hydraulic Research, 37:6, 743-757. doi: 10.1080/00221689909498509*

Tassi, P., (2017). *Sisyphe User Manual.*

Thapa, B. (2004). *Sand erosion in hydraulic machinery. Norwegian University of Science and Technology, Faculty of Engineering Science and Technology.*

Villaret, C., Hervouet, J., Kopmann, R., Merkel, U., Davies A. G. (2011). *Morphodynamic modeling using the Telemac finite-element system.*

White, R. (2001). *Evacuation of sediments from reservoirs.*

Wilber, D. H., and Clarke, D. G. (2001) *Biological effects of suspended sediments: A review of suspended sediment impacts on fish and shellfish with relation to dredging activities in estuaries. North American Journal of Fisheries Management, 21:4, 855-875.*

Yen, C., and Lee, K. T. (1995). *Bed topography and sediment sorting in channel bend with unsteady flow.*

Links of the theoretical guides, manuals

ANSYS Fluent (2009): http://users.ugent.be/~mvbelleg/flug-12-0.pdf

MIKE 21/3 (2017): https://www.mikepoweredbydhi.com/download/product-documentation

SSIIM (2014): http://folk.ntnu.no/nilsol/ssiim/

TELEMAC (Bourban, S., 2014): http://www.opentelemac.org/index.php/presentation