**Breugem, W. Alexander**

# Wetting and drying improvements in TELEMAC (part 1)

# Wetting and drying improvements in TELEMAC (part 1)

W.Alexander Breugem[1]
abr@imdc.be, Antwerp, Belgium
[1]: IMDC (NV)

*Abstract* – **This paper addresses wetting and drying using the TELEMAC finite element code in two-dimensional and three-dimensional simulations. Simulations with wetting and drying in TELEMAC can be challenging, especially in situations where the bottom slope is large. Here, high velocity values are regularly encountered, that sometimes can lead to instabilities and crashes of the model and often limit the time step of the simulations, thus increasing the calculation time. In this paper the TELEMAC finite element wetting and drying algorithm is studied. Some alternative methods for the wetting and drying scheme are implemented, which are tested in theoretical test cases as well as in a simplified version of the Scheldt Estuary model.**

***Keywords***: TELEMAC-2D, TELEMAC-3D, wetting-drying, tidal flats, numerical methods

## I. INTRODUCTION

### A. *Background*

In many simulations, the model domain consists of shallow areas, where a part of the time the area becomes dry. Examples of such dry areas include tidal flats, beaches, and riverbanks. The simulation of wetting and drying is numerically and physically challenging. The physical challenge lays in the fact that the shallow water equations, such as they are used in TELEMAC-2D are not valid, as some of the assumptions used in deriving these equations are not satisfied around the transition to dry areas. The numerical challenges in the wetting and drying process are many. The most important of these are:

- Ensuring that negative water depths do not occur.
- The occurrence of spurious pressure gradients on slopes (Figure 1).
- Ensuring correct local and global mass balances (Figure 2).
- The occurrence of shocks at the wet-dry transition. In TELEMAC, this often leads to the occurrence of wiggles in the velocity, with peaks that are unphysically large.
- The occurrence of singularities due to division by the water depth (e.g. in the bed friction term and the horizontal diffusion term).
- Occurrence of instabilities leading to model crashes. In TELEMAC, these typically happen because high velocities occur on (nearly) dry cells.
  Preventing this typically leads to a severe time step criterion and therefore slow calculations. It seems in general, that this problem tends to be more severe close to (steep) slopes in the bathymetry.
- Model crashes due the occurrence of NaN values in the tracers (like temperature or water quality variables) in shallow areas.

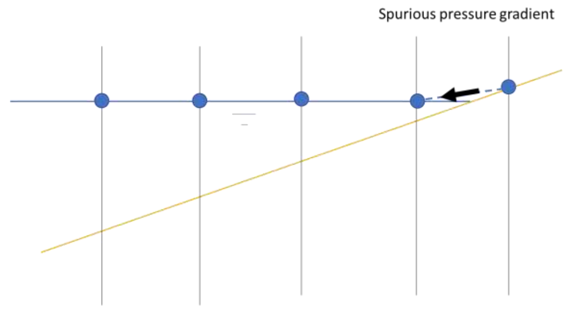


Figure 1. Occurrence of spurious pressure gradients due to wetting drying. Because the water depth is non-negative, the free surface on the dry slope has an incorrect gradient, which drives a spurious flow when left uncorrected.

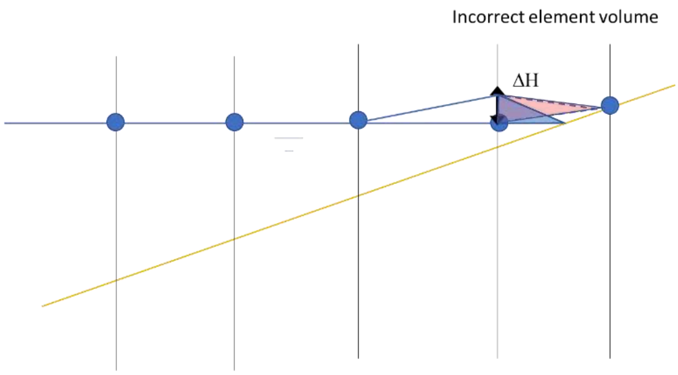


Figure 2. Occurrence of mass balance errors due to wetting drying. The volume that needs to be filled/released on the dry area is larger in the model (red) than it should be in reality (blue), leading to increased flow on the side.

It is interesting to note that the first two problems could disappear in case negative water depths are allowed. This approach is indeed used in the finite element model of Henische et al [1], who apply a very large friction coefficient in areas with negative water depths in order to damp the flow there. Nevertheless, this approach has many limitations, and hence, is not considered in this paper.

### B. *Objective of the study*

The objective of this research is to further improve the wetting and drying algorithm in TELEMAC. In particular, the aim is to make the algorithm more stable, such that larger time steps can be used, and to limit artifacts (wiggles) in the computed velocity fields, which pose problems for example when sediment transport is considered.

In this paper, the focus is to find solutions that work in TELEMAC-2D and TELEMAC-3D for cases with and without transport of scalars. This means that many numerical options are not considered in the present work, because they are not available in TELEMAC-3D (such as finite volume schemes, the primitive equation option or SUPG on water levels) or because they cannot be used in combination with tracer transport (such as filtering to correct negative depths: TREATMENT OF NEGATIVE DEPTHS = 1). Note that the finite volume method in TELEMAC-2D, while certainly more robust for wetting and drying, has a stringent time step criterium, because it is an explicit scheme, making it rather slow.

These considerations lead us to the choice to inspect the case where the wave equation is used to solve the momentum equations (TREATMENT OF THE LINEAR SYSTEM = 2), in combination with the flux control settings to treat negative depths (TREATEMENT OF NEGATIVE DEPTHS = 2). In the present paper, only simulations are performed using TELEMAC-2D. Improvements in TELEMAC-3D are left to a future paper. Improvements to scalar transport are also not considered in this paper, but these are mentioned in [2].

### *C. Overview of wetting drying in literature*

It is insightful to consider the scientific literature on wetting drying. Most of the work on wetting and drying has been done in finite volume schemes. However, some work has been performed for wetting and drying. An implicit wetting-drying method was presented by Kärmä [3]. In their paper, they change the bathymetric elevation in order to prevent negative depths. The resulting system of equations is non-linear and is therefore solved using a Newton-Raphson solver. Wetting and drying in a finite element simulation for non-hydrostatic simulation is also considered in [4] and [5], whereas the solution of the shallow water using residual distribution schemes is described by [6]. In the latter approach, the scheme is constructed in such way that the water depth cannot become negative. Their approach has some similarities to the negative depth algorithm in TELEMAC, which also uses a residual distribution scheme.

Stelling and Duinmeijer [7], whose approach was later extended to unstructured meshes [8], consider wetting and drying on structured finite difference scheme, and present an implicit scheme for the continuity equation, which still retains a time step limit (only one cell can be flooded per time step), which is needed to make sure the water depth remains positive. Their conclusion is that for correct wetting and drying, an energy balance (obtained using Bernoulli's equation) should be applied on the transition of wet to dry. This shows that the characteristics method used for advection of velocities (such as used in TELEMAC) leads to additional energy dissipation. Hence it is a robust choice, but it may not be suited for all circumstances.

Casulli [9] presented a new finite volume method to take wetting and drying into account. His approach considers the variation in the bathymetry within a cell of the mesh. This has two advantages: the water depth of the volume fluxes is better represented (typically they are deeper, leading to less formation of shock waves), and it solves the volume and pressure gradient problems discussed in section I.A. Because the method uses bathymetry variation within cells, simulations can be performed using coarser meshes while keeping the same accuracy, thus leading to faster calculation times. The method leads to a non-linear system of equations, which is solved using a Newton-Raphson iteration technique that was found to converge rapidly. Unfortunately, it is not straightforward to apply these ideas to the finite element method used in TELEMAC.

### *D. Overview of wetting drying algorithm in TELEMAC*

The algorithm that is used in TELEMAC-2D and TELEMAC-3D to calculate the hydrodynamics is roughly summarized as follows:

1. First, boundary conditions, forcings and source terms (baroclinic pressure gradient, Coriolis force, wave-current interaction etc.) for the hydrodynamic equations are calculated.
2. The changes in the water level and velocities are calculated using the wave equation (subroutines propag.F in TELEMAC-2D and wave_equation.f in TELEMAC-3D). This is a matrix equation, which needs to be solved. The calculation consists of the following sub steps:
   a. Calculation of the vertical flow profile based on vertical diffusion and forces (TELEMAC-3D only).
   b. Calculation of the advection terms (these are explicit).
   c. Calculation of the horizontal diffusion. This is done using an explicit method in TELEMAC-3D, and a somewhat implicit method in TELEMAC-2D[1].
   d. Calculation of the vertical momentum equation to determine the non-hydrostatic pressure component (TELEMAC-3D, non-hydrostatic calculations only).
   e. Calculation of the free surface gradient at time *n*. A correction is applied to eliminate the spurious pressure gradient (Figure 1).
   f. Calculation of an auxiliary velocity, based on source terms, advection and diffusion as well as the surface pressure gradient at step *n* (that does not yet include the effect of changes in the water level).
   g. Solving the wave equation to determine the changes in water levels. At this step a matrix equation is solved.
   h. Applying the changes in water level to determine the water level gradient at time *n*+q, and use this gradient to calculate the final flow velocity.

[1] In TELEMAC-2D the diagonal of the diffusion matrix is applied implicitly, where the off-diagonal terms are applied explicitly. This has the advantage that no extra linear system needs to be solved, while still being stable for large values of $DDT/DX^2$. Here D is the diffusion coefficient, DT the time step and DX a measure for the mesh size. However, the calculated diffusion term leads to diffusion fluxes that are too low, especially for large values of $DDT/DX^2$.

3. Determine the volume fluxes of water (subroutine flux_ef_vf.f).
4. Recalculate the water levels using the volume fluxes from step 3 using a residual distribution scheme (subroutine correction_depth_2d.f). This scheme is globally mass conservative, meaning that there is no loss of the mass of water, and at the same time, it prevents the occurrence of negative water depths. The scheme iteratively distributes the water masses that are transported between nodes [10]. In case the algorithm finds that no more water can be redistributed, it stops iterating. Typically, this happens when the flux out of a cell during a time step is larger than the available volume of water (i.e. during drying). A known issue of this scheme is that the results may differ when the number of parallel processes is changed. This issue is not addressed in the present paper.

The advantage of the algorithm used in TELEMAC is that a layer of water (although with a thickness of 0 m) always remains present, thus avoiding instabilities related to including or excluding elements from the calculation. The fact that the algorithm allows the water depth to become zero is also an advantage. In many models, the remaining water layer needs to have a minimum thickness, which leads to problems with the volume balance of water, that can be severe in case large tidal flats are present (e.g. [11]). Finally, the algorithm is implicit, so it should, in theory, not pose any time step criterion.

## II. Overview of Test Cases

### A. Introduction

The tests are performed using the goblinshark branch, which is based on TELEMAC v8p1. However, this branch contains two important changes with respect to wetting and drying:

- The velocities are set to zero at dry areas (defined as $h < 0.01$ cm), as it was found that this prevents many instabilities, thus permitting the use of relativly large time steps.
- Forces in the momentum equation (such as Coriolis force) are set to zero below a threshold depth ($h < 0.10$ m), after finding crashes related to the application of the force in shallow areas when using the NERD scheme for advection of momentum in TELEMAC-3D.

### B. Thacker Fruit bowl

#### 1) Description of the test

The test case described by Thacker [12] is used as the main test case to test wetting and drying. In this test case, a seiche is calculated in a circular domain, where the bottom has a parabolic variation. Thacker showed that this situation has an analytical solution for cases without any energy dissipation (bottom friction, and viscosity), but including the advection of momentum as well as the Coriolis force.

In this study, a circular domain with a radius of 10,000 m is set up with a mesh size of 100 m. This lead to a mesh with 34,261 nodes (67,787 elements). The bathymetry is chosen such that the water depth in the middle of the basin is equal to 5 m. As an initial condition, the water level from the analytical solution is used for the moment the water level has the most extreme run up to the right (Figure 3). At that moment, the velocities are zero according to the analytical solution, so zero velocities were used as an initial condition.

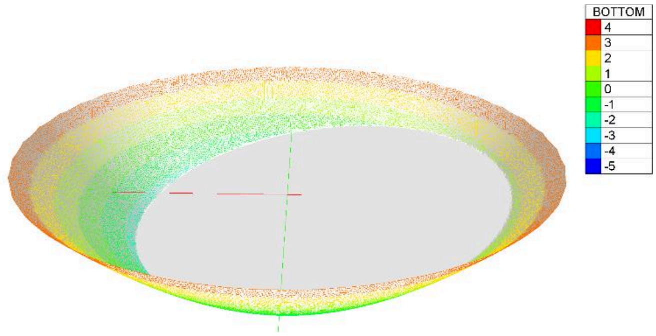


Figure 3. Mesh and bathymetry of the test case, and initial condition of the water level (in grey).

The simulations are performed for a total duration of 12 h, which corresponds to roughly nine oscillation periods. The time step is set to 30 s. The settings for the model were taken as much as possible in accordance to the analytical solution (for the case with a Coriolis coefficient of $f = 0$), with one major exception: bed friction is used applying Nikuradse's law with a friction coefficient of $k = 0.003$ m, which is relatively low. This was done, in order to be able to assess the effect of bed friction, as this is sometimes considered an essential process for the correct numerical simulation of wetting and drying. For advection of momentum, the NERD scheme (14) is used.

In each test case, it is tested that the oscillation period from the simulations corresponds to the value calculated in the analytical solution (80 minutes) and that the mass of water is conserved. Only when differences are found herein, this will be mentioned in the text. Note further that the analytical solution gives a maximum velocity of 0.8 m/s. Any velocity higher than this value is considered a spurious artifact.

#### 2) Results and sensitivity analysis of Thacker fruit bowl

The simulations show that using the finite element method, two problems occur (note that according to the analytical solution, the velocity is expected to be constant in the domain):

- The occurrence of wiggles in the velocity with a rather large magnitude. These occur near the wetting front. (Figure 4 at a distance of -8 km).
- The occurrence of a discontinuity in the velocity, around the drying front (Figure 4 at a distance of +8 km). Apparently, the drying does not occur fast enough in the model, leading to the lagging of the drying front.

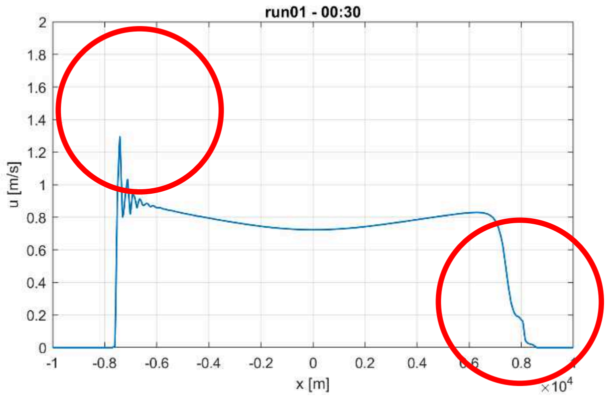


Figure 4. Velocity magnitude after 30 min on a transect through the centre of the basin. The flow at this moment is from right to left.

A sensitivity test is performed, studying a large amount of numerical and physical parameters, in order to understand and to find, whether some numerical parameters can solve the issue. The main conclusions of this sensitivity study are:

- The wiggles are strongly influenced by the time step. Decreasing the time step diminishes the wiggles (but even with a very small time step of 0.5 s, they do not disappear completely. This is in contrast to the finite volume method, which does not show any wiggles. However, the finite volume method is slow (about a factor 10 slower than the base case with a time step of 30 s), due to the fact that it uses a very small time step of approximately 1.5 s.
- Using a high horizontal diffusivity (20 m2/s), diminishes the wiggles. However, it has a strong side effect, namely that the oscillation period changes substantially (with 10% to 88.5 s). Further, there is a stability limit, thus limiting the applicability of this method in case with fine meshes.2
- Some numerical parameters lead to a substantial decrease of the wiggles. The most prominent ones are NUMBER OF SUB-ITERATIONS FOR NON-LINEARITIES = 3, although this leads to a substantially larger calculation time (it increased almost with a factor 3), and FREE SURFACE GRADIENT COMPATIBILITY = 0.
- There is some influence of the advection scheme. Nevertheless, the results using the characteristics method instead of the NERD scheme are rather similar. Other advection schemes have not yet been tested.

### *C. Scheldt test case*

#### *1)* Description of the test

A field test case is also taken. For this, an extract is made of the upstream branches of IMDC's Scheldt model [13], in order to have a fast model that can easily be run on a single processor. The mesh contains 40,658 nodes (68,230 elements) and contains channel meshes in a large part of the domain (Figure 5). The area contains many narrow branches, often discretised using channel meshes, with small mesh sizes (of the order of 10 to 30 m in the streamwise direction and up to 5 m in the spanwise direction). The bathymetry in many of these branches has rather steep slopes to the side, on which wetting and drying occurs. This makes it a challenging test case for the numerical scheme.

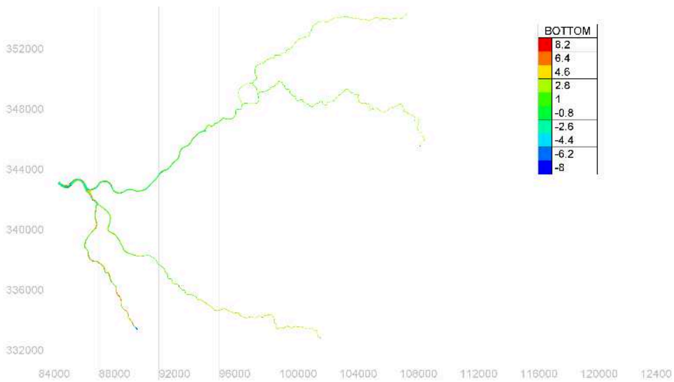


Figure 5. Mesh and bathymetry of the Scheldt test case.

As boundary conditions, measured flow rates are used upstream, whereas a time series of water levels extracted from the full Scheldt model is used at the downstream boundary.

The simulations are performed using TELEMAC-2D with a time step of 30 s. This is a very large time step, but one of the objectives of the test is to increase the time step as much as possible in order to obtain the fastest simulation time. It was found that the fastest simulations on a single processor were obtained using the direct solver (8), whereas simulations using the conjugate gradient method (1) take longer, as more than 500 iterations are needed for the solver to converge. This shows that for very large time steps, the direct solver can be a fast alternative for the iterative solvers (at least in serial mode). This also suggest that for simulations with very large time steps, a speed-up might be obtained using more advanced preconditioning than is currently available in TELEMAC.

#### *2)* Results and sensitivity analysis of the Scheldt test case

The model runs even with the large time step. However, relatively high velocities are found (O 10 m/s). These are related to spurious velocities generated on nearly dry areas. A limited sensitivity analysis was performed. It is found that increasing the diffusivity, though showing a tendency to smooth out some shock waves, also rapidly leads to instabilities and model crashes, meaning that setting the diffusion coefficient alone cannot solve drying flooding issues in this case. Moreover, it is found that flow occurred through closed walls (Figure 6). At the location of the closed wall, there are vectors visible with components perpendicular to the wall. Probably, these are generated due to the free surface slope, which is quite steep in that direction. The condition that $\nabla\vec{u}.\vec{n} = 0$ at the closed boundaries, is apparently not fulfilled automatically Therefore, a correction is implemented, which is very similar to the routine airwik2.f in TELEMAC-3D, in which the velocity at the boundary is corrected in order to ensure that $\nabla\vec{u}.\vec{n} = 0$ at the closed boundaries. The results are shown in Figure 7. It is clear that the flow through the boundary is stopped. Further, it appears that the maximum

[2] Indeed increasing the horizontal viscosity in the Scheldt test case let rapidly to instabilities and crashes.

velocities decrease in that case (closer to physical plausible values) and that the flow field is somewhat smoother. Nevertheless, the flow field in Figure 7 clearly has issues, in the sense that there is high flow from the (nearly) dry banks into the river.

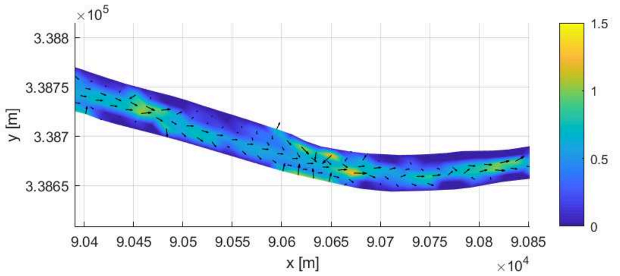


Figure 6. Instantaneous velocity field without boundary correction.

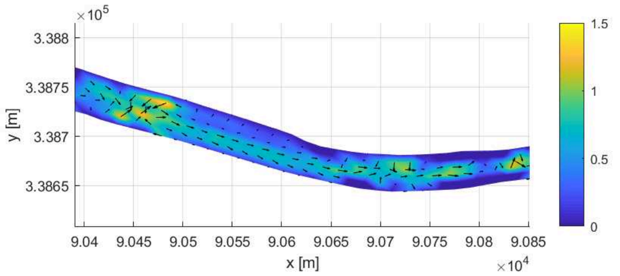


Figure 7. Instantaneous velocity field after correcting the boundary.

## III. ALTERNATIVE WETTING DRYING METHODS

### A. *Alternative depth for propagation*

From the parameter analysis, it is found that one of the parameters that had an impact on the formation of wiggles was NUMBER OF SUB-ITERATIONS FOR NON-LINEARITIES. This suggests that the wiggles might be mitigated by having a better value of $h_{\{prop\}}$, the water depth that is used in the continuity equation for the calculation of the flux. Indeed, it is remarked [9] that using deeper values for the propagation depth prevents the formation of shocks. However, using sub iterations is slow (roughly three times slower for 3 sub-iterations). Therefore, an alternative method is used here, namely to extrapolate $h_{\{prop\}}$ using the change in the water level at the previous time step $\Delta h^{n-1}$:

$$h_{prop} = h^n + \theta_H(h^n - h^{n-1}) = h^n + \theta_H \Delta h^{n-1}$$

Here, $\theta_H$ is the implicitation factor for the water depth. The results are shown in Figure 8. As an alternative method, $h_{prop}$ is determined using the characteristics method from the water depth at time n, the results of which are shown in Figure 9 for the Thacker example. This method is inspired by finite volume methods, where the fluxes are often determined at an upwind location for greater stability.

The wiggles clearly decrease in both cases. The decrease is substantially stronger when using extrapolation. The reason may be that flow velocity was used in the characteristics method in order to determine $h_{prop}$, whereas it might be more physical to use the wave propagation velocity ($\sqrt{ch}$). No negative side effects are found in both cases with respect to mass conservation, change in the oscillation period or calculation time. Note that in both figures, the velocity discontinuity at the drying front does not change substantially, suggesting that the cause of this is unrelated to the calculation of the propagation velocity.

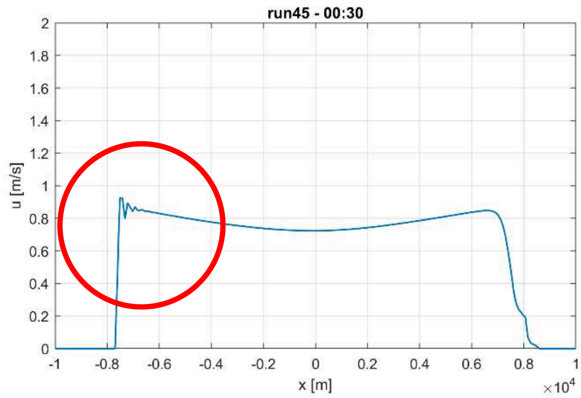


Figure 8. Thacker: Velocity magnitude after 30 min on a transect through the centre of the basin, with $h_{\{prop\}}$ calculated using extrapolation with $\theta_H$=1. The flow at this moment is from right to left.

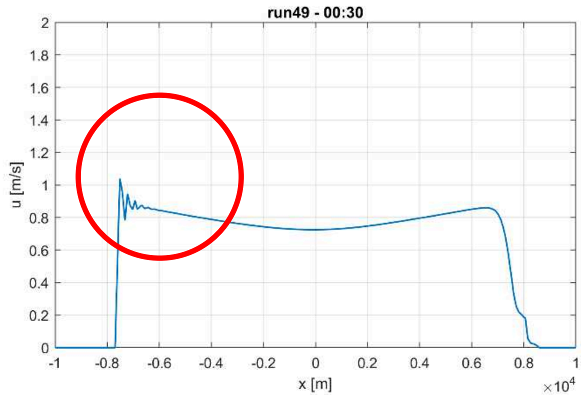


Figure 9. Thacker: Velocity magnitude after 30 min on a transect through the centre of the basin with $h_{\{prop\}}$ calculated using the method of characteristics. The flow at this moment is from right to left.

In the Scheldt test case, clear fronts at wetting areas are not found (temporary dry banks are more important here). Therefore, it is expected that little gain will be obtained from this method. This is to be investigated in future.

### B. *Velocity filtering*

As there are clear issues with the velocity field, an attempt has been made to explicitly filter out these wiggles. Thereto a family of filters is implemented, similar as in Wolfram and Fringe [14], where higher order filters are constructed by consecutively applying first order (smoothing) filters. The higher order filters should in principle filter higher frequencies, while not affecting lower frequencies (thus leading to less energy dissipation). Some numeric experiments showed that for the Thacker test case (which has few low frequency components, as the flow is essentially constant in space) the consecutive application of the first order is most effective, and therefore results of the higher order filter are not presented in this paper. The results are shown in Figure 10 and Figure 11. These figures show that one filter pass already limits the wiggles substantially. Because the water levels are recalculated using a residual distribution scheme that is perfectly conservative, the filter can be applied without affecting the global mass balance. Indeed, in the test, mass is conserved.

Applying more filter steps, leads to a further decrease of the wiggles and also smoothes out the erroneous flow on the right side of the bowl. Interestingly, the smoothing also decreases the wiggles in the free surface to some extent (not shown). However, this method is dangerous, as it can smooth out some flow features that one might be interested in (like eddies behind a bridge pillar). Therefore, it should be applied with care and is certainly not a solution for all cases. More tests are needed to investigate how strong this effect is.

Note that the increase in calculation time was rather limited, even applying the first order filter six times only leads to an increase in calculation time of 5%.

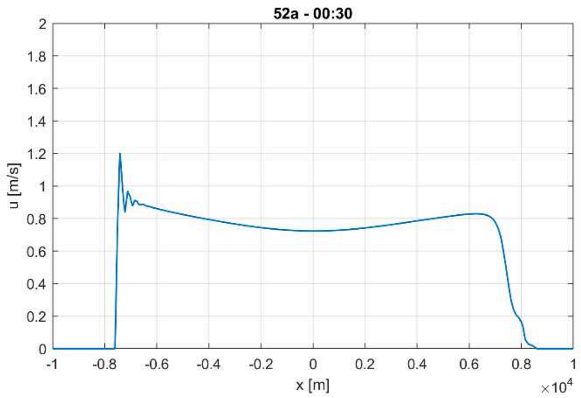


Figure 10. Thacker: Velocity magnitude after 30 min on a transect through the centre of the basin, using one iteration of a first order filter. The flow at this moment is from right to left.

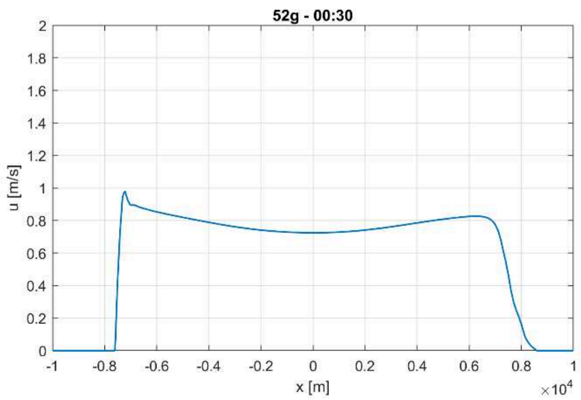


Figure 11. Thacker: Velocity magnitude after 30 min on a transect through the centre of the basin, using six iterations of a first order filter. The flow at this moment is from right to left.

Application of the filtering in the Scheldt test case gives interesting results Figure 12). Due to the use of the filter, all noise in the velocity field has disappeared and the flow is smooth and aligned in the direction of the river branch. However, this leads to a change in the tidal signal in this branch. A comparison with data will need to be performed in a later stage to see whether this is an improvement.

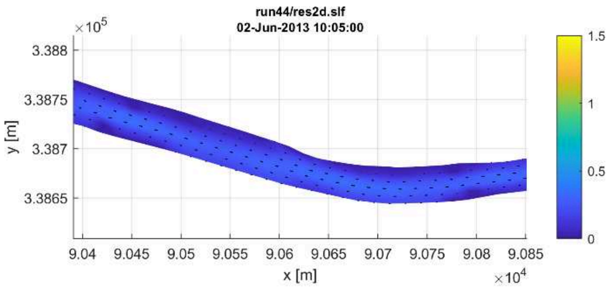


Figure 12. Scheldt: Instantaneous velocity field using five passes of a second order filter.

### C. *Alternative pressure gradient calculation*

In the shallow water equations, the free surface gradient drives the flow. With a semi-implicit discretization, this is given by:

$$\frac{u^{n+1} - u^n}{\Delta t} = \theta_u g \nabla \eta^{n+1} + (1 - \theta_u) g \nabla \eta^n + \cdots$$

Here, $\eta$ is the free surface elevation. In TELEMAC, the following expression is applied to write the free surface gradient as function of the change in water depth $\Delta H$:

$$\begin{aligned}\theta_u \nabla \eta^{n+1} + (1-\theta_u)\nabla\eta^n &= \theta_u \nabla(\eta^n + \Delta \mathrm{h}) + (1-\theta_u)\nabla \eta^n \\ &= \nabla\eta^n + \theta_u \nabla \Delta \mathrm{h}\end{aligned}$$

Then, with the option, OPTION FOR THE TREATMENT OF TIDAL FLATS = 1 or 3, a correction is applied to $\nabla\eta^n$ in order to mitigate the pressure level gradient problem (Figure 1). We have two hypotheses how the wetting drying in TELEMAC-2D could be improved:

1. The correction that is applied to calculate the free surface gradient is not physically correct and may lead to instabilities.
2. The correction for tidal flats is applied only to $\nabla\eta^n$, whereas it should be applied to $\nabla\eta^{n+\theta_H}$, with $\theta_H$ the implicitation factor for the water depth. It seems that it is tacitly assumed in TELEMAC that $\Delta$h always leads to a free surface profile at time step $n + \theta_H$, which does not have any issues with regard to wetting and drying. However, this may not be the case. This issue is subject to future investigations.

We will first look in more details to the free surface correction in TELEMAC-2D. In TELEMAC-2D, the water levels are corrected (in the routine corrsl.f) comparing the bathymetry with the free surface elevation. When the water level in the node with the highest bed level of an element is too high, it is decreased, whereas in the node with the lowest bed level of the element, it is increased up to the bed level of the node, which has a bed level between the two other nodes of the triangle. The water level in the middle node is left unchanged.

An example of the application of this algorithm is shown in Figure 13, where the free surface elevations before and after the correction are shown for a relatively smooth slope (3:1000) for a constant water depth of 0.1 m and a mesh size of 100 m. This example clearly shows the problems of the algorithm:

1. The free surface gradient is corrected even though it should not be. However, this typically leads to a decrease in the free surface gradient, hence lower velocities and a more stable model.
2. Whether the free surface is corrected depends on water depth and mesh resolution.
3. The correction creates larger free surface gradients in different directions in some elements.
4. For meshes and bathymetries where two nodes in an element have the same bed elevation (like studied here), the resulting corrected free surface depends on randomness/rounding errors in the input.

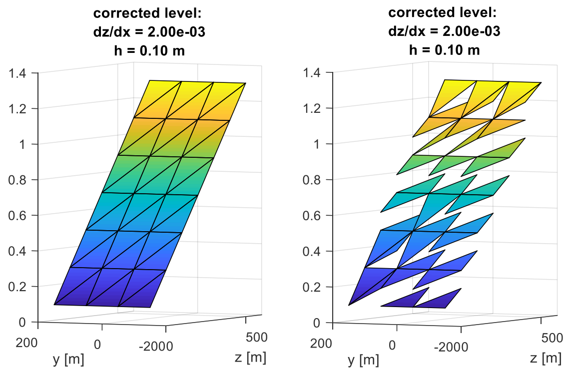


Figure 13. Illustration of the pressure gradient correction algorithm in TELEMAC. On the left, a 3D view of the original water levels is presented for a channel mesh with a constant bed slope and water level slope. On the right the corrected free surface elevation is shown, which is used to calculate the free surface gradient.

Note that the dependence on the resolution is also mentioned by Hervouet [15] (p.123) who states: "In certain cases (large elements along the slope and low water depth), this criterion is false and declares as dry elements that are entirely wet. This limitation should be taken into account while building the mesh. It should be ensured that the difference in level on an element is less than the depth". This is however a rather stringent criterion, which complicates the mesh making process hugely.

In order to assess the relation between water depth and free surface gradient, the average error in the free surface gradient was determined for a large number of meshes and water depths and summarized in Figure 14. Here, we see that the pressure gradient in the x direction decreases to almost zero for low water depths. Especially for coarse meshes, the water depth at which this happens is quite large. It is also clear that the error in the perpendicular direction can be substantial (i.e. of the same order of the error in the direction of the pressure). This is worrying, because the water level slopes in typical free surface flows are often rather small (of the order of 10-4 for typical lowland rivers as the Danube of the Rhine and even an order of magnitude for tides in coastal seas as the North Sea), which means that in principle, the pressure gradient should be determined rather accurately.

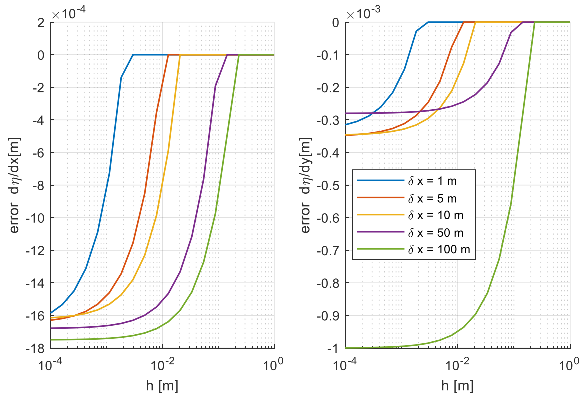


Figure 14. Synthetically determined error in the pressure gradient of TELEMAC's pressure correction algorithm as function of water depth and mesh resolution for a channel mesh. Left, error in the direction of the free surface gradient. Right: error perpendicular to the direction of the gradient.

An alternative method [6] for the calculation of the pressure gradient is tested here . In this method, the maximum bed elevation in an element is compared to the maximum water level of the wet nodes (using a threshold depth of 1.0 mm), and the water level is decreased for these dry points in order to decrease the gradient. The advantage of this method is that it does not artificially limit slopes when there is a certain water depth (such as happens in original free surface correction method in TELEMAC). The disadvantage of this method is that one has to apply a threshold depth, which specifies when the pressure gradient is modified. Note that this threshold does not have any influence on mass conservation. The results of this simulation are shown in Figure 15.The application of this correction shows limited difference with respect to the wiggles in the wetting front (statistics show it has become slightly worse). However, this equation shows a better velocity profile at the drying front. The reason is that the free surface gradient is not artificially limited, as happens when using the original free surface correction method TELEMAC. This leads to a calmer flow from the shallow areas and a smoother velocity profile.

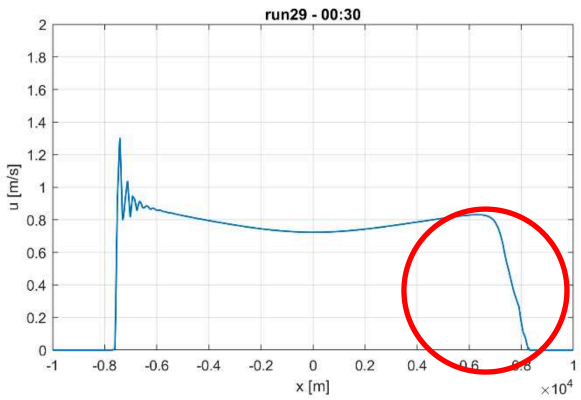


Figure 15. Thacker: Velocity magnitude after 30 min on a transect through the centre of the basin using the modified free surface gradient correction of Ricchiuto and Bollermann [6]. The flow at this moment is from right to left.

Application of the correction in the Scheldt model however, yielded disappointing results. High velocities were generated on the riverbanks, which led to high fluxes from shallow areas into the river. The negative depth algorithm struggled to cope with this. Hence it appears that the algorithm used in TELEMAC has a stabilizing effect by artificially lowering the free surface gradient in shallow areas.

### D. *Alternative bed friction*

Often, additional energy dissipation is used in models to prevent instabilities during wetting drying e.g. [4 and 5], who used increased bed friction and horizontal diffusion in shallow areas. Also in TELEMAC-2D, the bed friction is increased artificially using the maximum of the wave speed, and the flow velocity in the calculation of the bed friction for water depths lower than 0.03 m, (see the routine fricti.f).

Bi and Toorman [16] reason that in shallow areas, the flow becomes laminar, and includes this in the equation for the bed friction, and claims that this improves wetting drying. Based on their work, we propose the following simplified term for the friction drag $c_f$ , which was derived from their equation, neglecting the term containing the molecular viscosity[3]:

$$c_f = \frac{2}{\log\left(11.36\frac{h}{k}\right)} + \left(\frac{0.1k}{h}\right)^2$$

Here $h$ is the water depth and $k$ the Nikuradse roughness length. Basically, the bed friction is increased with an extra term that scales as $h^{-2}$, so this term increases rapidly for very small water depths. Note that laminar flow on slopes (where the most severe issues with wetting drying occur in TELEMAC) only occurs for very small water depths. For example, Breugem [17] measured a fully turbulent velocity profile in a water depth of only 0.016 m for a smooth bottom with a bed slope of less than 1%.

[3] Some analytical tests were performed in calculating the velocity on a constant slope for a constant water depth with and without this term. It was found that the differences were very small, and only became noticeable for small roughness values ($k$ = 0.003 m) and very small water depth ($H$ = 0.001 m).

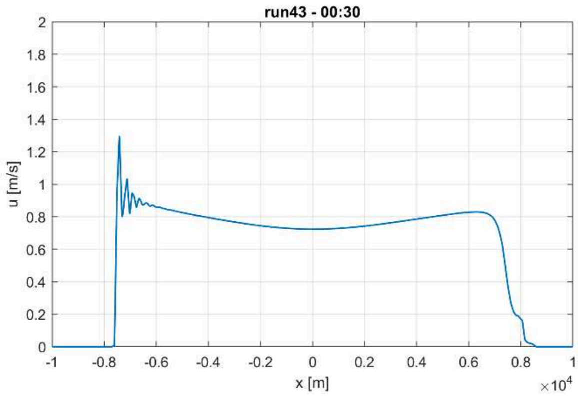


Figure 16. ghacker: Velocity magnitude after 30 min on a transect through the centre of the basin using the modified bed friction according to Bi and Toorman [16]. The flow at this moment is from right to left.

Applying the equation to the Thacker test case (Figure 16), we find extremely limited differences with respect to the occurrence of wiggles as well as with the velocity artifacts in the drying region. This is not very surprising. The increased bed friction only changes the drag coefficient for very low water depths (O (1cm)), whereas the problems in this test occur typically in deeper water depths. Note that in the areas where drying and flooding occur, the change in bed elevation per element is approximately 20 cm, meaning that the increased friction only occurs at areas that are already dry. Given these results it is concluded that the use of increased friction has a limited effect in this considered case, and hence more research to friction parametrisations do not seem the way forward.

Application of this friction law in the Scheldt test case confirmed this. Also in that case, the differences with the base case were very limited.

## IV. DISCUSSION

### A. *On thresholds for the minimum water depth*

Often, a threshold for the minimum water depth is used in shallow water models. Also, in TELEMAC, many of these thresholds are found. Examples include THRESHOLD FOR VISCOSITY CORRECTION ON TIDAL FLATS in TELEMAC-3D and THRESHOLD DEPTH FOR WIND in both TELEMAC-2D and TELEMAC-3D. Also, in this paper a threshold depth was used below which no flow occurs (section II.A).

Ideally, one would like to make these thresholds as small as possible. Further, one would like to have a physical base for these thresholds. An obvious lower limit for such a threshold based on physics would be the free molecular path of water, below which the continuum hypothesis, on which the shallow water equations are based, is not valid. However, this gives a minimum value around 2.5 $10^{-10}$ m, which is a value too low to be of practical use.

Another parameter that comes to mind to base the threshold depth on are the roughness length (Nikuradse length scale $k$)[4]. For situations with $h < k$, the flow is through the roughness

[4] I am indebted to Uwe Merkel for this suggestion.

elements rather than over it. Hence, in this situation, the flow would be largely blocked.

A third possibility is to use the sub grid variation in the bathymetry. A water depth that is lower than this variation is likely to be blocked by these flow features.

*B. Pressure gradient correction*

The pressure gradient correction used in TELEMAC was shown to underestimate the pressure gradient on slopes for coarse resolutions. This resolution dependence is a clear disadvantage of the method and should be taken into account when making meshes. The underestimations lead to a slowdown of the wetting on slopes as well as erroneous pressure gradients (even with wrong directions). An alternative pressure gradient correction algorithm was shown to solve this issue. However, in field cases with steep slopes in the bathymetry, it appeared that the algorithm in TELEMAC is rather stable (more so than the alternative one). The algorithms that were used here are relatively basic, but were not able to solve all problems, therefore, a more advanced algorithm seems necessary. This is the subject of further research.

*C. Negative depth algorithm*

The NERD scheme that is used in TELEMAC to make sure the water depths remain positive is a residual distribution scheme [10; see also section I.C]. First the volume of water $F_{ij}$ is calculated that needs to be distributed between nodes by:

$$F_{ij} = \Delta T \int \vec{u}.h_{prop} \nabla \psi_i d\omega$$

Here, $\psi$ is the basis function, and $\Delta T$ is the time step. Then this mass is iteratively distributed from upstream and downstream nodes along an edge. The iteration is stopped when either all the volumes to redistribute has become zero, or when the volumes to redistribute do not change anymore. In this case the fluxes are considered unphysical. Some simulations were performed (both in the Thacker basin and in the Scheldt test case) to find the locations where the fluxes did not converge to zero, and it was found that this happened normally in areas where drying occurs. This means that the non-physical fluxes happen when the flux calculated from the water depth and velocity from wave equation, is too large. In other words, there is not enough water available at these locations. Ideally, one would decrease the flow velocity in these areas in such a way that the flux decreases to a point where no fluxes remain in the redistribution method. However, it is not very obvious how this could be implemented. Instead, another option is to correct the velocities and water depth calculated using the wave equation in such a way that the available volume of water is taken into account. In order to do so, a type of iterative method seems necessary.

*D. Iteration for non-linearities*

The wetting drying process is an inherently non-linear process. Therefore, for the ultimate wetting drying method, it is needed to solve a non-linear system of equations, using an iterative solution method. Different non-linearities should be taken into account. Xia and Jiang [18] argue that at a wetting-drying front, there needs to be an equilibrium between bed friction and pressure gradient, which can only be obtained correctly if the friction term (*FRIC*) is discretised fully implicitly:

$$FRIC = \frac{c_d |u^{n+1}| u^{n+1}}{h^n},$$

whereas in most flow models (including TELEMAC), this term is linearised:

$$FRIC = \frac{c_d |u^n| u^{n+1}}{h^n}$$

Further, the importance of a correct representation of $h_{\{prop\}}$ (section III.A) also leads to a non-linearity, that should in principle be solved using iterations, even though extrapolation yielded quite good results. Further, the subgrid method [9], which solves the pressure gradient issue as well as the negative depth problem or Kärma's [3] method for correcting the free surface pressure gradient, leads to a system of non-linear equations.

In TELEMAC, iterations for non-linearities are present, but the implemented method has two disadvantages. First, it uses a fixed number of iterations, which sometimes leads to a high accuracy, and sometimes to a rather low one. Further it is rather slow, as the iteration occurs on the full hydrodynamic calculation.

Therefore, a start was made to implement an additional method to treat the non-linearities, where the first non-linearity that was implemented is the one related to the bed friction term. An accuracy threshold is used, in order to make sure the accuracy of the sub-iterations is always the same. This was done in a loop, containing steps 2f to 2h from section I.D, i.e. without recalculating the diffusion matrix and advection of momentum. This method is substantially faster than the original method (the calculation time increased by "only 50%" for a calculation with between 3 and 7 iterations), compared to a three times higher calculation time for 3 iterations in the original method. It is expected that a further speed-up might be obtained by using better precondition of the matrices and using better methods for dealing with the non-linearities (for example a quasi-Newton method). It is the intention to include step by step more non-linear processes and make more processes implicit (also some not related to wetting-drying like an implicit discretization of the Coriolis force, or some boundary conditions). This will be the building block for an ultimate wetting-drying method, which solves both the spurious pressure gradient and the mass balance method, and can be used in combination with the existing treatment of non-linearities, where the existing method functions similar to the outer iteration in a CFD code, and the new iteration method as an inner iteration.

## V. CONCLUSION

In this paper, the wetting-drying scheme for the finite element calculations in TELEMAC is analysed using a schematic test case of a seiche in a parabolic basin (Thacker) and a small model of a part of the Scheldt Estuary. It is shown that there are multiple problems related to the wetting-drying process. The first issue is the occurrence of wiggles. It was shown that these can be limited using either explicit filtering or using a better estimate of the depth for propagation. Further, the algorithm that is used for correcting the spurious pressure gradient has a resolution dependency, which can easily lead to an underestimation of the pressure gradient, and hence drying that happens too slowly. Alternative methods can solve this, but may lead to unphysical high velocities on nearly dry areas for steep bed slopes. Thus, a more advanced method for the calculation of the pressure gradient is necessary. It is also shown that alternative friction calculations (taking into account the occurrence of laminar instead of turbulent flow in very shallow areas) do not bring any improvement to the wetting drying modelling in the presented test case. Finally, ideas are presented how further improvements in wetting and drying can be obtained. Apart from that, it is needed to perform more test cases on wetting and drying. Also, the test in the current paper were performed in serial mode. Testing these features in parallel is still needed.

## REFERENCES

[1] Heniche, M., Secretan, Y., Boudreau, P., & Leclerc, M. (2000). A two-dimensional finite element drying-wetting shallow water model for rivers and estuaries. Advances in Water Wesources, 23(4), 359-372.

[2] Kyrousi, F. Chu. K,, Breugem, W.A. Decrop, B. & Gillis L.G. (2022) Water Quality modelling of the Southern North sea using TELEMAC-3D coupled with AED2, In preparation.

[3] Kärnä, T., Legat, V., & Deleersnijder, E. (2013). A baroclinic discontinuous Galerkin finite element model for coastal flows. Ocean Modelling, 61, 1-20.

[4] Funke, S. W., Pain, C. C., Kramer, S. C., & Piggott, M. D. (2011). A wetting and drying algorithm with a combined pressure/free-surface formulation for non-hydrostatic models. Advances in Water Wesources, 34(11), 1483-1495.

[5] Candy, A. S. (2017). An implicit wetting and drying approach for non-hydrostatic baroclinic flows in high aspect ratio domains. Advances in Water Resources, 102, 188-205.

[6] Ricchiuto, M., & Bollermann, A. (2009). Stabilized residual distribution for shallow water simulations. Journal of Computational Physics, 228(4), 1071-1115.

[7] Stelling, G. S., & Duinmeijer, S. P. A. (2003). A staggered conservative scheme for every Froude number in rapidly varied shallow water flows. International Journal for Numerical Methods in Fluids, 43(12), 1329-1354.

[8] Kramer, S. C., & Stelling, G. S. (2008). A conservative unstructured scheme for rapidly varied flows. International Journal for Numerical Methods in Fluids, 58(2), 183-212.

[9] Casulli, V. (2009). A high-resolution wetting and drying algorithm for free-surface hydrodynamics. International Journal for Numerical Methods in Fluids, 60(4), 391-408.

[10] Hervouet, J. M., Pavan, S., & Ricchiuto, M. (2017). Residual distribution advection schemes in Telemac RESEARCH REPORT N° 9087 August 2017 Project-Team CARDAMOM

[11] Balzano, A. (1998). Evaluation of methods for numerical simulation of wetting and drying in shallow water flow models. Coastal Engineering, 34(1-2), 83-107.

[12] Thacker, W. C. (1981). Some exact solutions to the nonlinear shallow-water wave equations. Journal of Fluid Mechanics, 107, 499-508.

[13] Wolf, T., Breugem, W. A., Chu, K., Decrop, B., Van Holland, G., Plancke, Y., & Stark, J. (2021). Sediment transport modelling (TELEMAC-3D + GAIA) case study: sand disposals in the Western Scheldt. In Proceedings of the papers submitted to the 2020 TELEMAC-MASCARET User Conference October 2021 (49-57).

[14] Wolfram, P. J., & Fringer, O. B. (2013). Mitigating horizontal divergence “checker-board” oscillations on unstructured triangular C-grids for nonlinear hydrostatic and nonhydrostatic flows. Ocean Modelling, 69, 64-78.

[15] Hervouet, J. M. (2007). Hydrodynamics of free surface flows: modelling with the finite element method. John Wiley & Sons.

[16] Bi, Q., & Toorman, E. A. (2015). Mixed-sediment transport modelling in Scheldt estuary with a physics-based bottom friction law. Ocean Dynamics, 65(4), 555-587.

[17] Breugem, W. A. (2012). Transport of suspended particles in turbulent open channel flows. PhD thesis.

[18] Xia, X., & Liang, Q. (2018). A new efficient implicit scheme for discretising the stiff friction terms in the shallow water equations. Advances in Water Wesources, 117, 87-97.