POLITECNICO MILANO 1863

SCUOLA DI INGEGNERIA INDUSTRIALE E DELL'INFORMAZIONE

Executive Summary of the Thesis

# Particle Finite Element Method applied to sediment transport and erodible surfaces

Laurea Magistrale in Mathematical Engineering - Ingegneria Matematica

**Author:** Simone Martini

**Advisor:** Prof. Alberto Corigliano

**Co-advisor:** Prof. Massimiliano Cremonesi

**Academic year:** 2022-2023

## 1. Introduction

The aim of this thesis is to study the main effects of flooding events on the nearby territory causing erosion and land damages. The urgency was brought out by the alluviums in the Marche region on the 15th and 16th of September 2022, in which there were 12 casualties, 50 injured and 150 people displaced. The total monetary loss of this disaster was around 2 Bilion of Euros. With this work, the hope is to have set the basis to develop a model which can be useful to predict the risk and consequences of particular events that can be harmful. The starting approach is the Particle Finite Element Method (PFEM), which describes the fluid domain with a Lagrangian mesh that follows the particles of the fluid. The task is to integrate this model with a description of the sediment movement inside the flow. In the literature, the most accurate way to approach the problem is considering a two phase flow. This is computationally very onerous and can create some problem in this PFEM environment since usually two phase flow models work with an Eulerian mesh for the fluid. So this way to approach the problem is discarded. Indeed the aim is to find something which is not computationally expensive but can give good results in some concrete applications. To this matter, a mixture model has been considered by adding an advection-diffusion equation to the solution system. From this equation the concentration of the sediment is recovered on the Lagrangian mesh of the fluid. This model has the limit of working with low concentrations of sediment inside the flow. Then, an erosion model based on the Shields scouring criterion is used to see how the flow affects the surrounding soil environment.

In this thesis, the total model is addressed considering first the mathematical setting of the problem, controlling that each part works in a robust way; then some concrete physical tests are carried on in order to find some possible improvements and to see the applications.

## 2. Fluid-sediment model

Consider a generic bounded, time-dependent domain $\Omega \subseteq \mathbb{R}^2$, $\forall t \in [0,T]$ with $T > 0$. In here, the Navier-Stokes equations for weakly compressible fluids are solved using the ALE (Arbitrary Lagrangian Eulerian) approach. The Eulerian formulation has been used to address the problem of boundary nodes. Indeed, they too are moving with the fluid and so they lose their position at the edge. To solve this issue, these boundary nodes are considered fixed and here the Eulerian formulation is used. Summing up, $\forall \mathbf{x} \in \Omega$ and $\forall t \in [0, T]$ the system to solve is:

$$\begin{cases} \rho \left[ \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u_c} \cdot \nabla) \mathbf{u} \right] = \nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{b} \\ \frac{\partial p}{\partial t} + (\mathbf{u_c} \cdot \nabla) p + K (\nabla \cdot \mathbf{u}) = 0 \end{cases} \tag{1}$$

where $\mathbf{u_c}(\mathbf{x}, t) = \mathbf{u}(\mathbf{x}, t) - \mathbf{r}(\mathbf{x}, t)$, with $\mathbf{r}(\mathbf{x}, t)$ denoting the mesh velocity. $\boldsymbol{\sigma}(\mathbf{x}, t)$ is the stress tensor, $\rho$ is the fluid density, $K$ is the fluid bulk modulus and $\mathbf{b}(\mathbf{x}, t)$ are the body forces. For the sediment the equation is:

$$\frac{\partial c}{\partial t} + \nabla \cdot \left[ \left( \mathbf{u}_c + w_s \frac{\mathbf{g}}{|\mathbf{g}|} \right) c \right] = \frac{\nu_t}{\sigma_c} \Delta c \tag{2}$$

Where $w_s$ is the falling velocity constant that addresses the effect of gravity, $\nu_t$ is the sediment diffusivity and $\sigma_c$ is the Schmidt number. Gravity is present as an advection term since in this starting model the sediment does not have mass and so no body forces can be present. For now a weakly coupled model is considered: this implies that the concentration is influenced by the fluid velocity but not vice versa. In the next part, a totally coupled model is introduced as an improvement. For the moment the density of the fluid is constant and so is the viscosity. The generic boundary condition for the Navier-Stokes (NS) equations and for the advection-diffusion (AD) one are:

$$NS = \begin{cases} \boldsymbol{\sigma} \cdot \mathbf{n} = 0 & \text{on } \Gamma_N \\ \mathbf{u} = \mathbf{q} & \text{on } \Gamma_D \end{cases} \tag{3}$$

$$AD = \begin{cases} \nabla c \cdot \mathbf{n} = 0 & \text{on } \Gamma_N \\ c = w & \text{on } \Gamma_D \end{cases} \tag{4}$$

Note that $\Gamma_D \cup \Gamma_N = \partial\Omega$ and $\Gamma_D \cap \Gamma_N = \varnothing$. $\mathbf{n}$ is the normal vector to the boundary and $(\mathbf{q}, w)$ are the imposed values of velocity and concentration at $\Gamma_D$. Finally, an initial condition is considered for velocity and for concentration such that, $\forall \mathbf{x} \in \Omega$:

$$\begin{aligned} \mathbf{u}(\mathbf{x}, 0) &= \overline{\mathbf{u}} \\ c(\mathbf{x}, 0) &= \overline{c} \end{aligned} \tag{5}$$

## 2.1. Numerical formulation

In order to be able to solve the problem, a mesh discretization in space $\Upsilon_h$ and a partition in time have been adopted. $\mathbb{P}^1$ linear functions are used to represent velocity, pressure and concentration. This creates some problems regarding the satisfaction of the inf-sup condition and so a proper stabilization method should be implemented, which in our case is the pressure stabilizing Petrov-Galerkin technique. The temporal discretization of the equation is the explicit Eulerian one where all the terms are considered at the previous instant. The time subdivision used is:

$$\begin{gathered} 0 = t^0 < t^1 < ... < t^n < ... < t^M = T \\ t^{n+1} - t^n = \Delta t^n, \, n = 0, 1, ..., M-1 \end{gathered} \tag{6}$$

which introduces this particular time stepping procedure to solve the set of equations.

**Step 1: Solve the momentum equation to find the velocity $\mathbf{u}^{n+1}$**

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^n}{\Delta t^n} + (\mathbf{u}_c^n \cdot \nabla) \mathbf{u}^n = \frac{1}{\rho} \nabla \cdot \boldsymbol{\sigma}^n + \mathbf{b} \tag{7}$$

**Step 2: Update the position of the mesh nodes**

$$\mathbf{x}^{n+1} = \mathbf{x}^n + \mathbf{u}^{n+1} \Delta t^n \tag{8}$$

**Step 3: Solve the advection-diffusion equation to find the concentration of the sediment $\mathbf{c}^{n+1}$**

$$\begin{aligned} \frac{c^{n+1} - c^n}{\Delta t^n} + \nabla \cdot \left[ \left( \mathbf{u}_c^{n+1} + w_s \frac{\mathbf{g}}{|\mathbf{g}|} \right) c^n \right] \\ = \frac{\nu_t}{\sigma_c} \Delta c^n \end{aligned} \tag{9}$$

**Step 4: Solve the continuity equation to recover the pressure $\mathbf{p}^{n+1}$**

$$\begin{aligned} \frac{p^{n+1} - p^n}{\Delta t^n} + \left( \mathbf{u}_c^{n+1} \cdot \nabla \right) p^n \\ + K \left( \nabla \cdot \mathbf{u}^{n+1} \right) = 0 \end{aligned} \tag{10}$$

Each solution has to fill the boundary condition set for the particular problem. They can be as seen for the cases 3 and 4. To address the stability issues of the explicit Euler method, the time step is fixed by following the CFL condition:

$$\Delta t^n = C_N \min_e \left( \frac{h_e^n}{v_e}, \frac{(h_e^n)^2}{\nu_t} \right) \tag{11}$$

where $h_e$ is a current characteristic size of the deformed element $e$, $v_e$ is the speed of dilational waves in the fluid depending on the element density and $C_N$ is a safety parameter. This condition has been modified since the sediment equation can give problems regarding the choice of the stable time step.

### 2.2. Totally coupled model

Now that the numerical setting of the model is done, the first limit is addressed. Indeed, for now the concentration did not have influence on the motion of the fluid. The aim is now shifted into finding a totally coupled model, where the fluid and the sediment interact with each other. In order to do this the density and the viscosity of the fluid are changed by inserting the concentration.

$$\rho = \rho_f (1 - c) + \rho_s c \tag{12}$$

$$\mu = \mu_f (1 + 2.5c) \tag{13}$$

where: $\rho_f$ is the fluid density, $\rho_s$ is the sediment one and $\mu_f$ is the fluid viscosity. The formula 13 is found in [2] and it works under the main assumption of $c \leq 0.1$. This limit is in line with the physical problem of the mixture model: it works only with low concentrations. This problematic has been explained in section 1, showing why in the PFEM framework it makes sense to use the mixture model.

## 3. Erosion model

Now the erosion model is introduced in the system by considering the Shields scouring criterion, as shown in [3]. The basic idea is to consider each node belonging to the interface and check if the shear stress applied by the fluid is enough to snatch it away from the bed. In this case the node becomes a part of the fluid flow. The shear stress is computed passing through the Shields parameter:

$$\theta = \frac{\tau}{(\rho_s - \rho)\, g D_m} = \frac{\rho \left\|\mathbf{u}^*\right\|^2}{(\rho_s - \rho)\, g D_m} \tag{14}$$

$$\theta_c = 0.22 R_{ep}^{-0.6} + 0.06\; 10^{-7.7 R_{ep}^{-0.6}} \tag{15}$$

$$R_{ep} = \frac{D_m \sqrt{D_m g \left( \frac{\rho_s}{\rho} - 1 \right)}}{\mu} \tag{16}$$

$D_m$ is the mean dimension of the sediment, $g$ is the acceleration of gravity, $\rho_s$ is the sediment density, $R_{ep}$ is the particle Reynolds number and $\mathbf{u}^*$ is the friction velocity. If $\theta > \theta_c$, the node is freed. The friction velocity is computed in the nearest node to considered one at the interface using the log law formula:

$$\mathbf{u}^* = \kappa \frac{\mathbf{u} - (\mathbf{u} \cdot \mathbf{n})\, \mathbf{n}}{\ln \left( \frac{z_\Delta}{z_0} \right)} \tag{17}$$

where $z_\Delta$ is the distance between the fluid node and the boundary, as shown in figure 1. To satisfy the log law profile assumption, it should be $30 \leq \frac{z_\Delta}{z_0} \leq 130$.

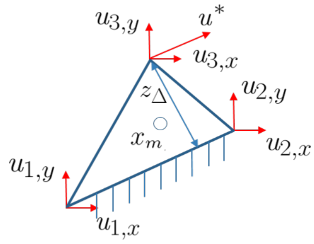


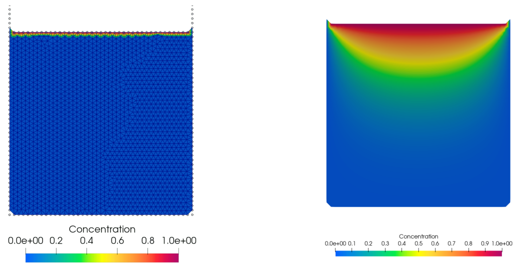

Figure 1: Boundary element: the friction velocity is computed using the velocities at the third node

## 4. Results - Flow model

To have a better understanding of each term in the sediment equation, they are added brick by brick. Initially, only the diffusion action is considered, then the transport one is added: in this way the full model will be built step by step giving a good overview of the meaning of each part. Finally, a case showing the effects of the totally coupled model is presented.

### 4.1. Diffusion

To study the diffusion part of the equation, $\mathbf{u} = 0\ \frac{m}{s}$ and $w_s = 0\ \frac{m}{s}$: the fluid is still and gravity is not acting on the sediment. The domain is a bucket on which a value $c = 1$ is imposed on the free surface. The starting domain and the final result are shown in figure 2.
An equilibrium condition is reached quickly, showing the fulfilment of the boundary conditions. To check the good functionality of this




(a) $t = 0\ s$ (b) $t = 1\ s$

Figure 2: Diffusion action: $\nu_t = 10\ \frac{m^2}{s}$, $\sigma_c = 1$, $h_m = 0.1\ m$

model, a convergence analysis has been carried on. The expectation was to recover the error estimate in $L^2(\Omega)$ of equation 18:

$$\|c - c_h\|_{L^2(\Omega)} \leq \overline{C} h^{r+1} |c|_{H^{r+1}(\Omega)} \tag{18}$$

where $c$ is the solution of the analytical problem, $c_h$ is the discretized one, $\overline{C}$ is a constant and $r$ is the grade of the polynomial used to discretize the solution, which in our case is 1. In figure 3, it can be seen that the error has a quadratic order of convergence in $h$, which follows the grade expected. Then, using the commercial software ABAQUS ©, a further confirmation of the good functioning of this model has been made.

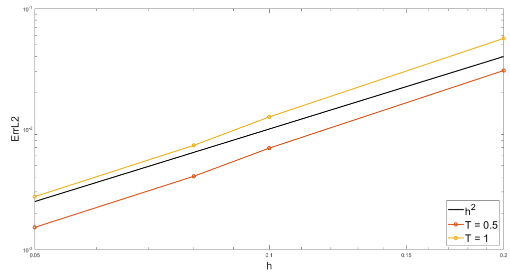



Figure 3: $L^2$ error estimate (log-log scale)

| $\|c - c_h\|_{L^2(\Omega)}$ | | | | |
|---|---|---|---|---|
| h[m] | 0.2 | 0.1 | 0.08 | 0.05 |
| $T = 0.5s$ | 0.03607 | 0.00694 | 0.00405 | 0.00152 |
| $T = 1s$ | 0.05656 | 0.01260 | 0.00731 | 0.00274 |

Table 1: Values of the $L^2$ errors

## 4.2. Advection

Now the transport terms are activated, by considering non null fluid and falling velocities. On the other hand the diffusivity of the sediment is set to be null, so $\nu_t = 0\ \frac{m^2}{s}$. The test considered for this case is a channel flow with a circular sediment source that is transported along by the fluid movement. In figure 4 is represented the starting system: the fluid moves from left to right and the velocity is null on the top and bottom; the sediment source is a circle that starts in the center. The results in figure 5 show that the sediment is going to the right (fluid transport) and down (gravity transport). After $T = 10\ s$ (final time) the source exits the domain.

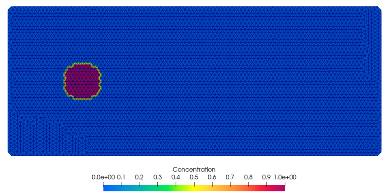



Figure 4: $t = 0\ s$, $w_s = 0.05\ \frac{m}{s}$, $h = 0.1\ m$

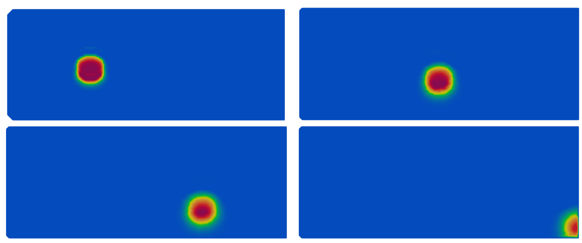


Figure 5: Concentration plot at: $t = 1s$, $t = 3s$, $t = 5s$ and $t = 8s$

This problem is advection dominated: a Streamline Diffusion stabilization has been introduced after having computed the local Peclet number:

$$\mathbb{P}e_k = 1.25 h_k \frac{\left\| \mathbf{u} + w_s \frac{\mathbf{g}}{|\mathbf{g}|} \right\|_{L^\infty} + \frac{\nu_t}{\sigma_c}}{\frac{\nu_t}{\sigma_c}} \tag{19}$$

Then, the global Peclet number is recovered as $\mathbb{P}e = \max_k \mathbb{P}e_k$. If $\mathbb{P}e > 1$, the local SD term 20 is added in the weak formulation:

$$\frac{h_k}{\|\boldsymbol{\beta}\|_{L^\infty}} \int_{\Omega_k} (\boldsymbol{\beta} \cdot \nabla c_h)(\boldsymbol{\beta} \cdot \nabla v_h)\ d\Omega \qquad \boldsymbol{\beta} = \mathbf{u} + w_s \frac{\mathbf{g}}{|\mathbf{g}|} \tag{20}$$

Practically, a diffusive term is added of the order of the transport term. In this way all the spurious oscillations go away. Avijit et al. [1] recover a convergence estimate for SD, which should be of order $\frac{3}{2}$ as shown in formula 21:

$$\|c - c_h\|_{L^2} \leq \frac{C}{\sqrt{\gamma}} h^{3/2} \tag{21}$$

where $C$ and $\gamma$ are constants. This convergence estimate is confirmed by the model as shown in figure 6.

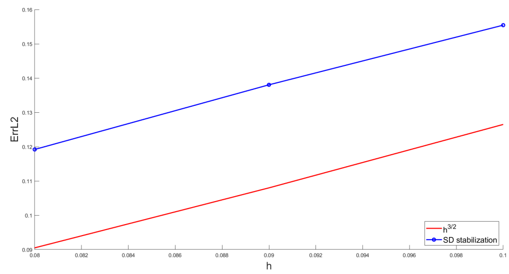


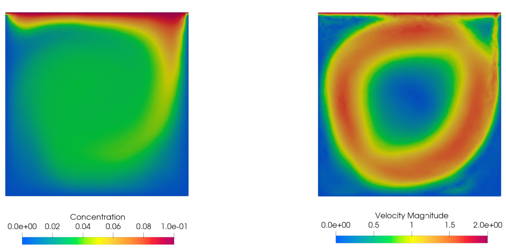

Figure 6: SD convergence (log-log scale)

| $\|c - c_h\|_{L^2(\Omega)}$, $T = 1s$ | | | |
|---|---|---|---|
| h[m] | 0.1 | 0.09 | 0.08 |
| SD-Stab | 0.15544 | 0.13804 | 0.11919 |

Table 2: Values of the $L^2$ errors for the stabilized problem

### 4.3. Totally coupled model

In this thesis many problems were attacked, showing different applications of the model. To test the correct functionality of the totally coupled model, a cavity flow is considered. Cavity flow is a problem where a fluid moves inside a box by forcing the velocity on the top boundary. Indeed, the velocity of the fluid and the concentration on all the sides are 0 except on the top one, where a positive horizontal velocity is imposed to the fluid and $c = 0.1$.




(a) Concentration plot (b) Velocity plot

Figure 7: $T = 60\ s$, $\nu_t = 0.01\ \frac{m^2}{s}$, $w_s = 0.05\ \frac{m}{s}$, $h = 0.02\ m$

The final solution can be viewed in figure 7 and the plot of the concentration in two points is shown in 8. The problem reaches a steady state situation, where the sediment (gravel, $\rho_s = 1680 \frac{kg}{m^3}$) is trapped inside the main vortex of fluid.

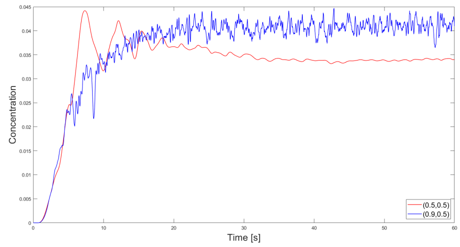



Figure 8: Time plot of the concentration in $(0.5, 0.5)\ m$ and in $(0.9, 0.5)\ m$

## 5. Results - Erosion model

To check if the added erosion model works fine, some concrete simulations were carried on. The most significant ones that will be shown here are the sand dune erosion and the beach erosion. These cases should set the starting point for what this model can build on in flood protection environment.

### 5.1. Sand dune erosion

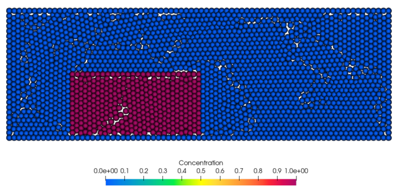



Figure 9: Starting domain of the problem

The domain is a channel of size $[0, 3] \times [0, 1]\ m$, shown in figure 9. As long as the dune particles (sand, $\rho_s = 1520 \frac{kg}{m^3}$) in $\Omega_d = [0.5, 1.5] \times (0, 0.5]\ m$ do not violate the Shields condition, they have the imposed values $c = 1$ and $\mathbf{u} = 0 \frac{m}{s}$. The boundary conditions for the fluid are the same as for the problem described in section 4.2. Regarding concentration, the natural condition is imposed at the exit while on all the other channel boundaries $c = 0$. The results are shown in figure 10, where it can be seen that the sand dune is extinguished in $T = 30s$. In figure 11 is instead shown the plot of the eroded area in

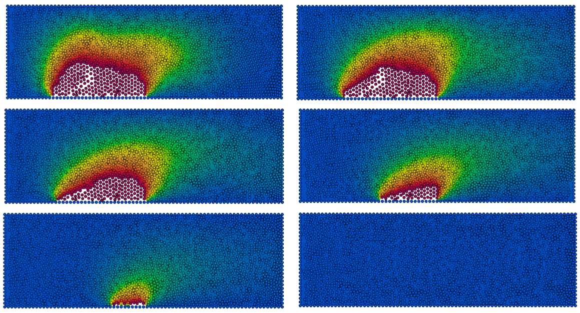


Figure 10: Concentration plot at: $t = 1s$, $t = 2s$, $t = 4s$, $t = 10s$, $t = 20s$ and $t = 30s$

time: at the start the erosion is more aggressive since in the center of the channel the velocity is bigger; then erosion is attenuated by the fact that the basis of the sand dune is near the bottom where the velocity is small due to the null boundary condition.

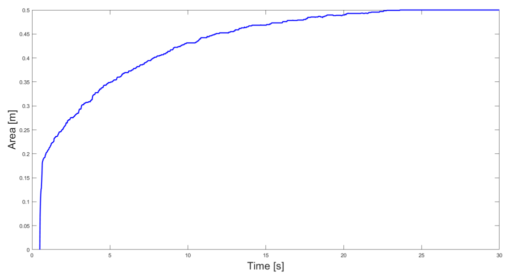



Figure 11: Plot of the eroded area

## 5.2. Beach erosion

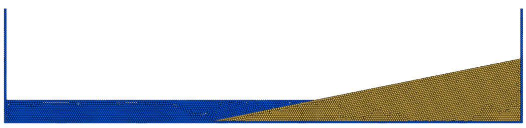
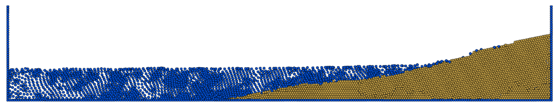


Figure 12: Starting domain of the problem

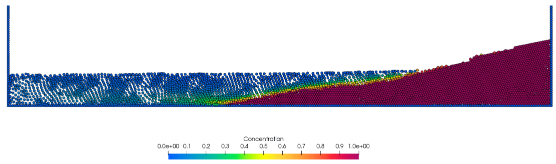
Now the focus is shifted on the beach erosion problem. In figure 12 the initial domain is shown, composed by the sea and the beach covering a total distance of 25 meters. At the start, the sea has no sediment inside; while the beach is composed only by sediment, indeed here $c = 1$ and the nodes are still. The sea is perturbed by a set of waves that shatter on the beach causing its erosion. Since the fluid must lose energy in order to snatch away the sediment particles from the beach, the totally coupled model is used. Thus, the loss of energy is addressed by the variability of density and viscosity. In this problem high concentrations must be considered: a better formula found in [4] for the mixture viscosity is considered in equation 22.

$$\frac{\mu}{\mu_w} = 1 + 2.5c + 10.05c^2 + 0.00273e^{16.6c} \quad (22)$$

This model works for concentrations $c \leq 0.4$, which is a better limit with respect to $c \leq 0.1$ but it is still not ideal. Viewing the results of figure 13, take into account that there is an error to keep in mind since for the beach nodes $c = 1$.



(a) Beach morphology



(b) Concentration plot

Figure 13: $T = 90\ s$, $\nu_t = 0.001\ \frac{m^2}{s}$, $w_s = 0.001\ \frac{m}{s}$, $h = 0.1\ m$

The beach is eroded by the waves and the released sand precipitates on the bottom of the sea, since the concentration is higher towards the seabed.

# 6. Conclusions

The final objective of this thesis was to build an accurate, robust and computationally affordable model for the simulation of sediment transport and erosion in a fluid-sediment domain. The aim has been reached, making some assumptions and taking into account some limits. The main one is that this model works only by considering low concentrations, which is a forced choice due to the fact that two-phase approaches are avoided for their modelling and numerical complexity. The mixture model on the other hand, is very fast and computationally affordable. All the test cases described in this thesis can be applied in different fields of engineering. There are many possible improvements to this model: first of all, the 3D extension should be implemented; then the Robin boundary condition should be introduced, which can represent correctly the sedimentation of the particles on the bottom surface due to gravity; finally, the Exner equation could be implemented, that describes bed morphology change due to erosion and deposit of sediment.

## References

[1] D. Avijit and S. Natesan. Convergence analysis of a fully-discrete fem for singularly perturbed two-parameter parabolic pde. *Mathematics and Computers in Simulation 197*, pages 185–206, 2022.

[2] A. Einstein. A new determination of molecular dimensions. *Ann. Phys. 19*, pages 289–306, 1906.

[3] N. Galano, P.A. Moreno-Casas, and J.A. Abell. Extending the particle finite element method for sediment transport simulation. *Comput. Methods Appl. Mech. Engrg. 380*, pages 1–16, 2021.

[4] Richard M. Iverson. The physics of debris flows. *Reviews of Geophysics*, pages 1–52, 1997.