LES two-phase modelling of suspended sediment transport using a two-way coupled Euler-Lagrangian approach

Daniel Wildt YP[1,2*], Michael Tritthart[2]

[1]Christian Doppler Laboratory for Sediment Research and Management, University of Natural Resources and Life Sciences, Vienna, Austria; [2]University of Natural Resources and Life Sciences, Vienna, Austria

*Correspondence YP: daniel.wildt@boku.ac.at

Abstract: A two-phase LES model using Euler-Lagrangian two-way coupling has been developed in OpenFOAM. The model shall be used for the analysis of the interaction between particles and their surrounding fluid. Validation of the model has been performed using data from physical experiments..

Keywords: suspended sediment, computational fluid dynamics, large eddy simulation, Euler-Lagrangian two-way coupling.

Methods

The development of a sediment plume in turbulent channel flow is studied using a two-phase LES model. Fluid flow is governed by the three-dimensional Navier-Stokes equations. The equations are solved on a structured mesh with 5.2 million cells (Yücesan et al., 2020). The movement of the dispersed phase (particles) is governed by Newton's law of motion and modelled using the Lagrangian approach. Particle diameters are uniformly distributed in the range of 0.50mm < d < 0.71mm. Momentum exchange between fluid and particles is two-way coupled. The entire model is solved using a modified version of the OpenFOAM-solver MPPICFoam (Greenshields, 2018).

Results

Worf et al. (2019) measured a fall velocity in the range of 0.03ms^{-1} to 0.06ms^{-1} for particles described above in turbulent channel flow with an average flow velocity of 0.5ms^{-1}. The fall velocity in the numerical simulation of this experiment was determined as 0.06ms^{-1} (Figure 2).

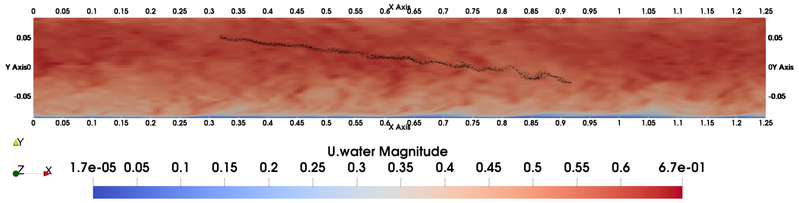

Figure 1: Sediment release into turbulent channel flow (length in m, velocity in ms-1); x is the direction of the bulk fluid velocity and the gravitational acceleration is pointing in negative y direction.

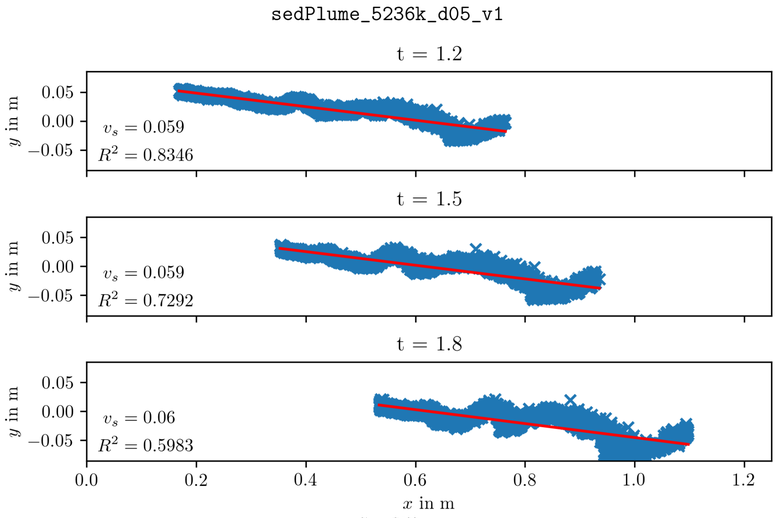

Figure 2: Particle fall velocity determination

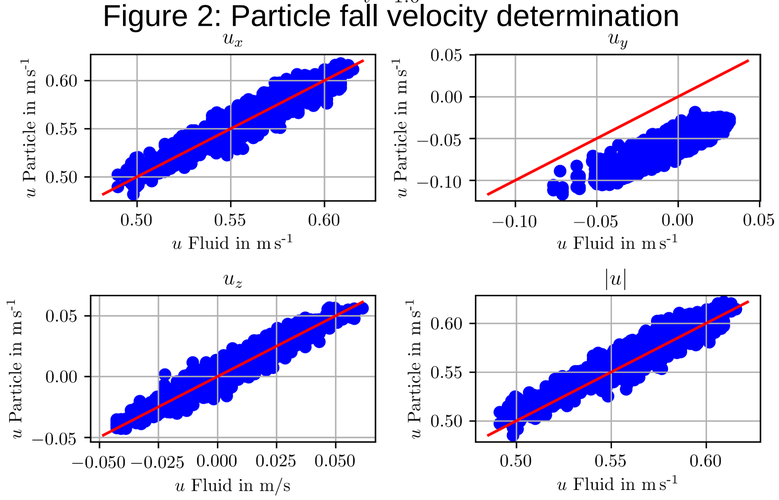

Figure 3: Comparison of the particle velocity and the velocity of its surrounding fluid 1 s after release into turbulent channel flow; each point represents one particle.

In Figure 3 the velocity of each individual particle is plotted against the velocity of its surrounding fluid. As a result of particle settling, the y component of the particle velocity is lower than the y component of the fluid velocity.

Conclusions

In this study the development of a sediment plume in turbulent channel flow has been numerically modelled and the velocity difference between particles and fluid has been analysed.

Bibliography

Greenshields C. J. (2018). OpenFOAM. The OpenFOAM Foundation. URL: http://openfoam.org.

Hauer C., Wagner B., Aigner J., Holzapfel P., Flo¨dl P., Liedermann M., Tritthart M., Sindelar C., Pulg U., Klo¨sch M., Haimann M., Donnum B. O., Stickler M., Habersack H. (2018). State of the art, shortcomings and future challenges for a sustainable sediment management in hydropower: A review. Renewable and Sustainable Energy Reviews 98, 40–55. doi:10.1016/j.rser.2018.08.031.

Worf D., Schobesberger J., Lichtneger P., Habersack H., Sindelar C. (2019). Sedimentation velocity of saualm crystalline using image processing methods. In 16th International Symposium on Water Management and Hydraulic Engineering.

Yücesan S., Schobesberger J., Sindelar C., Hauer C., Habersack H., Tritthart M. (2020). Large eddy simulation of a sediment particle under entrainment conditions. Journal of Hydraulic Research. revised.