Abstract Submitted
for the DFD20 Meeting of
The American Physical Society

Turbulence Modeling in Smoothed Particle Hydrodynamics FRANCESCO RICCI, New Jersey Institute of Technology, RENATO VACONDIO, Universit di Parma, ANGELANTONIO TAFUNI, New Jersey Institute of Technology — The continuous growth of computational power in Computational Fluid Dynamics (CFD) has made it possible to study flow at high Reynolds numbers, which demand reliable models for the simulation of turbulence. For Eulerian methods, this has led to a shifting from Reynolds Averaged Navier-Stokes (RANS) models to Large Eddy Simulation (LES), especially for industrial applications. Among Lagrangian approaches is Smoothed Particle Hydrodynamics (SPH), a meshless method often used to study free-surface flow in several fluids engineering problems. In the present work, the standard SPH method is compared with an Eulerian SPH model, the latter being a modification of the standard SPH approach in which the position of the particles is kept fixed and additional convective terms are added to the governing equations. This has allowed the identification of the main source of error for the standard SPH approach when simulating isotropic turbulence decay, i.e. the discretization error due to the irregular distribution of SPH particles. Such error leads to an inaccurate description of the decay of the kinetic energy and the turbulent structures in the flow. Different strategies to address these issues are then proposed and described.

Angelantonio Tafuni
New Jersey Institute of Technology

Date submitted: 10 Aug 2020 Electronic form version 1.4