Joel Peltier, L.; Knight, Kelly J.; Rosendall, Brigette; Pal, Sanjeeb; Rizhakov, Andri; Smirnov, Andrei; Iek, Chanthy

Computer simulation of liquid-solids slurries for wastewater treatment

Hydrolink

COMPUTER SIMULATION OF LIQUID-SOLIDS SLURRIES FOR WASTEWATER TREATMENT

BY L. JOEL PELTIER, KELLY J KNIGHT, BRIGETTE ROSENDALL, SANJEEB PAL, ANDRI RIZHAKOV, ANDREI SMIRNOV & CHANTHY IEK

Applied Simulation and Analysis; Bechtel Nuclear, Security, and Environment; Reston, Virginia 20190

Processing of liquid-solids slurries for wastewater treatment involves handling of dissolved solids and undissolved solids with readily suspended to rapidly settling behaviors. Given a significant loading of dissolved or readily suspended solids, the effective carrier-fluid rheology may exhibit complicated non-Newtonian effects. A simulation-based assessment of wastewater treatment requires a sophisticated computational fluid dynamics (CFD) flow code with submodels sufficient to address this potentially diverse range of physics. Reynolds-Averaged Navier-Stokes (RANS) models are the current workhorse.

Simulation is always limited by available computational resources and physics parameterizations. With advances in computational engineering in parallel processing environments and physics submodel development for computer simulation codes, many limitations are either being removed or are being moved to higher-order details. CFD-RANS models are now able to meet challenges for simulating liquid-solids slurry flows in complicated configurations.

Industrial wastewater may contain a significant fraction of undissolved solids with potentially broad particle size and density distributions. Granular-Eulerian multiphase modeling is an example of a CFD-RANS technology that has been formulated to handle this kind of application. In a Granular-Eulerian multiphase model, each gas, liquid, or solids constituent is treated

Figure 1. Multiphase mixing in an industrial process vessel: Flow (left), Velocities (Right)

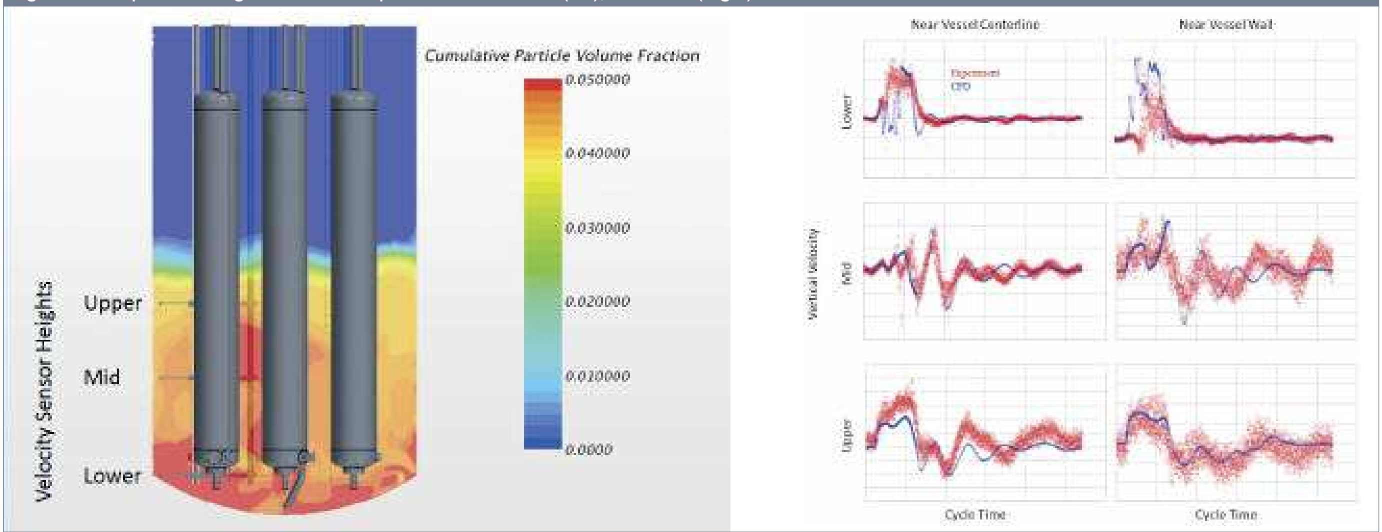
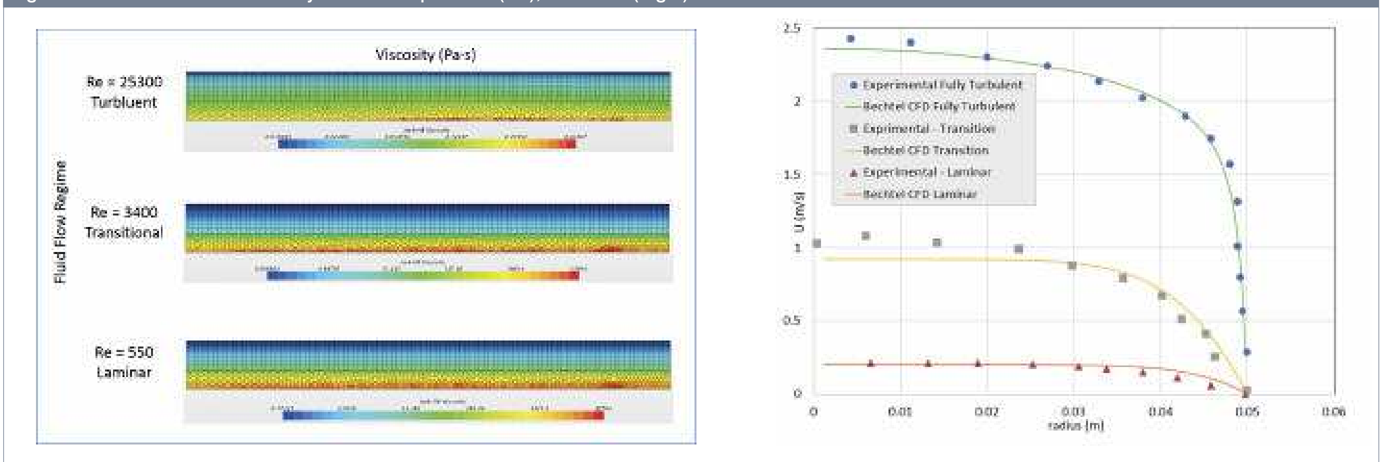


Figure 2. Flow of a Herschel-Bulkley Fluid in a Pipe: Flow (left), Velocities (Right)



as a separate continuous modeling phase. Submodels are used to parameterize interactions and behaviors at boundaries.

Comparisons of computational results (from a commercial CFD code CD-adapco/Star-CCM+) to experimental data show the fidelity that can be achieved. Figure 1 (left) is an instant from a simulation of mixing of a polydisperse loading of undissolved solids in a Newtonian carrier fluid. The mixing is performed in a vessel prior to the next step of the treatment process. The total solids loading in the vessel is 10% by weight. Approximately half of the solids are readily suspended. The upper part of the vessel is gas. The particle distribution is characterized by 6 solids phases with representative particle sizes and densities. The simulation presented in this article models an existing physical model experiment of the mixing of the waste in the vessel. The simulation geometry is derived from a CAD model of the experimental apparatus. In both the simulation and the experiment, time histories of velocity are sampled at six points in the bulk flow with the velocity sampling locations at lower, mid, and upper levels. Three locations provide velocities near the vessel centerline. Three locations provide velocities near the vessel outer wall. Comparisons of the CFD-RANS predicted velocities to the experimental data, Figure 1 (right), confirm model fidelity to real-world physics.

Dissolved and undissolved readily-suspended solids in industrial mixing vessels and other liquid-solids slurries may be modeled using an effective fluid rheology and density. Contemporary CFD solvers include a broad range of rheology submodels, a non-Newtonian Herschel-Bulkley fluid being an example.

In a Herschel-Bulkley fluid, the apparent viscosity of the fluid depends on the local shear rate. In regions of high local shear rates, a Herschel-Bulkley fluid behaves like a Newtonian fluid. As local shear rates reduce, a Herschel-Bulkley fluid becomes more viscous. The local shear rate in a turbulent flow occurs in the dissipation range of turbulence. CFD-RANS solutions provide energy-containing-range (mean-field) statistics, not dissipation range statistics. Without an appropriate model linking local shear rates to mean-field statistics, CFD simulations of Herschel-Bulkley fluids are well defined only for laminar flows where the dissipation range can be resolved explicitly.

A method to extend CFD modeling of Herschel-Bulkley fluids into the turbulence regime was recently presented at the Star-CCM+ Global Conference (Peltier et al, 2016). This model extension uses turbulence theory to estimate representative local maximum shear rate magnitudes from CFD-RANS data enabling simulations of Herschel-Bulkley fluids in the turbulence regime.

Figure 2 (left) shows CFD predicted viscosities for flow in a pipe of a Herschel-Bulkley fluid in the laminar, transitional, and turbulence regimes. The slice shown is from the pipe centerline to the upper outer wall. Comparisons of the CFD predicted velocities to experimental data, Figure 2 (right), confirm model fidelity to real-world physics.

The examples shown for simulation of liquid-solids slurries underscore that capabilities of contemporary commercial CFD flow codes are rapidly advancing and support a conclusion that a simulation-based assessment of wastewater treatment is possible with an expectation for fidelity to real-world physics. ■

L. Joel Peltier **is a Principal Engineer and manager of Computational Fluid Dynamics (CFD). He specializes in applied fluid mechanics, CFD, and numerical heat transfer applied to nuclear waste treatment and industrial processes. He is a member of ASME and Vice-Chair of the ASME Verification & Validation 20 Code and Standards Committee.**

Andri Rizhakov **is a Senior Engineer in Computational Fluid Dynamics. He specializes in nuclear waste treatment, multiphase flows for industrial applications, and analytics work via machine learning paradigms. Andri's leadership activities include team guidance involving writing of CFD quality process procedures. He is a member of the ASME.**

Kelly J Knight **is a Principal Engineer and manager of Advanced Simulation and Analysis (ASA). Kelly specializes in simulation of fluid and structural mechanics including modeling of blast protection, structural stress/strain and fatigue, multiphase flows, shock and blast wave events and thermal heat transfer. She is a member of the ASME.**

Andrei Smirnov **is a Senior Engineering Specialist in Computational Fluid Dynamics. He specializes in design of nuclear reactors, nuclear waste treatment, multi-phase flow analyses for industrial applications, and combustion software development. He is a member of the ASME and the ANS and a holder of one US patent.**

Brigette Rosendall **is a Computational Fluid Dynamics Specialist for Solar Turbines in San Diego. She specializes in solving problems involving multi-phase flow, combustion, radiation, species transport, free-surface, and moving boundaries. She is a member of the AIChE.**

Chanthy Iek **is an Engineering Specialist in Computational Fluid Dynamics. He specializes in shock wave dynamics and radiation/boiling/particulate deposition for chemical processes. His engineering experiences include CFD, turbomachinery, aero-acoustics, and air-breathing and rocket propulsion. He is a member of the ASME.**

Sanjeeb Pal **is a Senior Engineering Specialist in Computational Fluid Dynamics. He started working for Bechtel Corporation in 2008. His work in the ASA group is focused on Computational Fluid Dynamics application to address engineering challenges on Bechtel EPC Projects. He is a member of the ASME.**

References

Peltier, J., Rizhakov, A., Rosendall, B. Inkson, N., and Lo, S., 2015: Evaluation of RANS Modeling of Non-Newtonian Bingham Fluids in the Turbulence Regime using STAR-CCM+®, STAR-CCM+ Global Conference Location: San Diego, CA, March 15-18, 2015.