Postdoctoral Proposal

Numerical Simulations of Heat Transfer in Fluidized Beds
Application to Concentrated Solar Power Plants

Context

Circulating fluidized beds have exceptional transport and mixing characteristics. They notably benefit from the efficient contact between the dispersed phase and the gaseous medium, the thermal inertia of the particulate phase and excellent wall transfer capacities. These characteristics make them ubiquitous in the energy field, particularly for the development of innovative processes that meet the challenges of the energy transition. Gas-solid circulating fluidized beds are being studied at the PROMES laboratory as an alternative to current heat transfer fluids - which are used to transport heat obtained from solar radiation in concentrated solar power plants. This work has been carried out in particular by the European projects CSP2, Next-CSP and currently P2P [1 - 4] as well as by the ANR SicSun. In this process, solar radiation is concentrated on vertical tubes within which a gas-particle mixture circulates. Understanding and mastering the flow regimes involved, and the associated heat transfers, currently remain scientific obstacles for the development of this technology. The couplings between dynamics, thermals, the two-phase nature of the flow, wall effects and collisions make the physics particularly complex. To better understand these couplings, the PROMES laboratory is developing a multi-scale approach. At the solar receiver scale, Euler-Euler simulations are implemented with the Neptune_CFD software. At the local scale, we implement fine numerical methods where fluid-particle interactions are explicitly resolved. The high-performance computing (HPC) code TrioCFD based on a Front-Tracking method has been modified to allow the simulation of solid particles [5]. It has been successfully used to simulate fluidized bed flows with several thousand particles (see Figure 1) [6].

The analysis of technological barriers related to solar applications identifies as a priority the realization of simulations taking into account the combined transfers between the wall and the fluidized bed (gas/particle mixture). Unfortunately, the modeling of wall heat flows at the Euler-Euler scale is currently not capable of accurately predicting the heat transfers between the wall and the gas/particle mixture. The major scientific challenge concerns the study of the collective phenomena of particles and the identification of the effect of the complex shape of particles on hydrodynamics and heat transfers. These achievements, essential for the application to concentrated solar power plants, have never yet been carried out in the context of fine simulations of fluid-particle flow and constitute a major opening towards multi-physics coupling.

Work to be performed

Three main work tasks have been identified. The first task of the project consists of implementing high-performance calculations of a highly anisothermal fluidized bed with a very large number of particles (>10,000). The particles will be resolved using the Front-Tracking method of TrioCFD software. The simulation results will be used to study heat transfer at the wall and, in particular, to evaluate, as a function of fluidization velocities and particle shape, the heat flux distribution at the wall between the solid and fluid phases. The second task consists of performing Euler-Euler simulations using Neptune_CFD software. Particular attention will be paid to the modeling of wall phenomena and their effects on heat flux. The third task of the project consists of a detailed comparison and analysis of the results obtained at two different scales: resolved particles and the Euler-Euler approach. Upscaling modeling will be used to determine the best parameters of the Euler-Euler models and, where appropriate, propose improvements to these models.

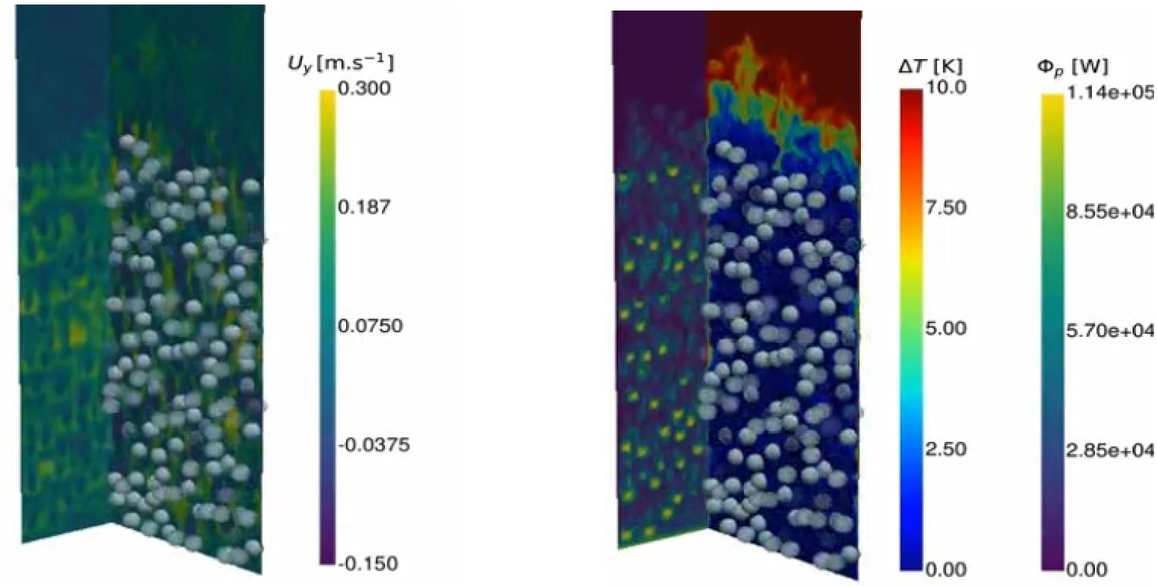

Figure 1: 3D HPC calculation (550 cores) of a fluidized bed of more than 2000 particles with 16 meshes per diameter [6]. Left: velocity field. Right: temperature field.

Duration and location: 1-year renewable PROMES-CNRS laboratory, Perpignan

Profile: We are looking for a PhD in computational fluid mechanics or energetics with a strong interest in modeling. Experience with HPC calculations and a programming language (C/C++/Python) is required.

Application

Adrien TOUTANT – 04 68 68 27 09 – adrien.toutant@univ-perp.fr
Samuel MER – 04 68 68 22 38 – samuel.mer@univ-perp.fr

Bibliography

[1] Next-CSP (2020). High Temperature Concentrated Solar Thermal Power Plant with Particle Receiver and Direct Thermal Storage. Available online: http://next-csp.eu/ (accessed on October 2022).

[2] Gueguen R., Sahuquet G., Mer S., Toutant A., Bataille F., Flamant G., Gas-Solid Flow in a Fluidized-Particle Tubular Solar Receiver: Off-Sun Experimental Flow Regimes Characterization, Energies, vol. 14(21), 7392, 2021.

[3] Sahuquet G., Gueguen R., Fontalvo L., Mer S., Toutant A., Bataille F., Flamant G., Particle Flow Heterogeneity in Fluidized-Particles Multitube Solar Receiver, SolarPACES, Albuquerque, USA, 27-30 September, 2022.

[4] Gueguen R., Grange B., Bataille F., Mer S., Flamant, G., “Shaping High Efficiency, High Temperature Cavity Tubular Solar Central Receivers”, *Energies*, 13, 4803. DOI: 10.3390/en13184803, 2020.

[5] Hamidi S., Toutant A., Mer S., Bataille F. Assessment of a coupled VOF-Front-Tracking/DEM method for simulating fluid-particles flows, International Journal of Multiphase Flow, vol. 165, 104467, 2023.

[6] Butaye E., Quintana R., Mer S., Bataille F., Toutant A., Investigation of heat transfers with particle-resolved simulations: From Stokes flow to fluidized bed, International Journal of Heat and Mass Transfer, vol. 241, 126687, 2025.