LABORATORY FOR CHEMICAL TECHNOLOGY
Technologiepark 125, 9052 Gent, Belgium

Coach	Supervisor(s)	Funding
Florian Wéry	Prof. dr. ir. Kevin Van Geem Prof. dr. ir. Geraldine Heynderickx	FWO

CFD-DEM investigation of a gas-solid vortex reactor geometry for the Oxidative Coupling of Methane

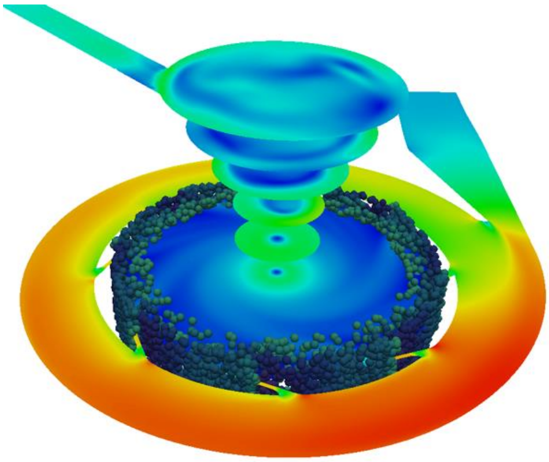
Aim

Investigate the applicability of the Euler-Lagrangian CFD-DEM technique on the GSVR by means of CFD simulations in the open-source package OpenFOAM coupled with LIGGGHTS.

Justification

The low natural gas price and the large amounts of shale and natural gas have created a renewed interest in methane as a source of liquid energy carriers or as a raw material for the chemical industry. One of the most promising processes to valorize methane into longer hydrocarbons is the catalytic oxidative coupling of methane (OCM). Two key challenges have to be addressed before OCM can be considered as an alternative gas-to-chemical technology, namely the low yields of ethylene and the efficient removal of the substantial heat release during reaction.

Catalytic processes inside fluidized bed reactors, known for their excellent heat transfer characteristics, are omni-present in the current chemical industry. Fluidization can take place in both the gravitational and centrifugal field. In industry, this is primarily performed inside the gravitational field. However, at the LCT, research is conducted on vortex reactors fluidizing catalyst particles in a centrifugal field. This allows for much higher gas throughputs compared to gravitational fluidized beds and a more narrow residence time distribution. For example, working in vortex reactors increases the control over the unwanted propagation of the gas-phase reactions and efficiently addresses the large amount of heat released during OCM operation.



CFD simulations allow for easy modification of the reactor geometry and operating conditions, optimizing the time-consuming experimental work flow. CFD-DEM combines Lagrangian particle tracking with an Eulerian approach for the continuous phase in order to capture the complex particle-particle and gas-particle interactions explicitly. These hydrodynamic CFD-DEM simulations will be performed with the CFD package OpenFOAM and DEM solver LIGGGHTS in this work.

Program

- Literature study on state-of-the-art CFD-DEM applications
- Gain acquaintance with the open-source CFD package OpenFOAM coupled with the DEM solver LIGGGHTS.
- Perform different cold and hot-flow hydrodynamic/reactive CFD studies, validated with experimental data.
- Optimization of design and operating conditions of the GSVR.

GHENT UNIVERSITY

LCT DRIVING CHEMICAL TECHNOLOGY