LABORATORY FOR CHEMICAL TECHNOLOGY
Technologiepark 125, 9052 Gent, Belgium

Coach	**Supervisor(s)**	**Funding**
Siyuan Chen Prof. dr. ir. Geraldine Heynderickx	Prof. dr. ir. Kevin Van Geem Prof. dr. ir. Yi Ouyang	/

CFD-DEM study of gas-solid hydrodynamics in a stator-rotor vortex chamber reactor

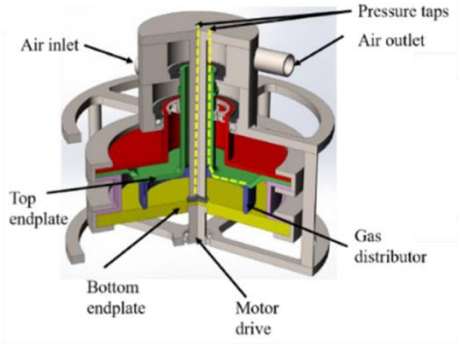
Aim

The aim of this thesis is to investigate the gas-solid hydrodynamics of a stator-rotor vortex chamber reactor using Euler-Lagrangian CFD-DEM simulation in the commercial software Ansys Fluent.

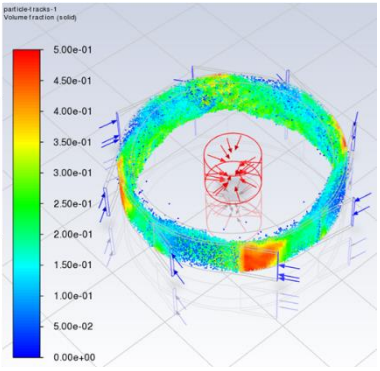
Justification

Process intensification (PI) is a promising tool to improve both reactive and non-reactive processes by increasing volumetric productivity, reducing energy consumption, and minimizing waste. The gas-solid vortex reactor (GSVR) has emerged as a promising intensified reactor technology due to its high heat and mass transfer rates[1].

However, the GSVR has some limitations, including a relatively high carrier gas consumption, high particle attrition due to particle-wall friction and a high risk of entrainment of fine powder. and An improved design of the GSVR, known as the stator-rotor vortex chamber (STARVOC), has been developed and patented by LCT. It features a rotor with several angled blades inside a static cylindrical chamber, which is driven by fluid's kinetic energy. The STARVOC technology has demonstrated a significantly improved energy efficiency, centrifugal acceleration, and solids loading[2]. To optimize the STARVOC design, such as finding the optimal geometrical parameters of the chamber, hydrodynamic studies are essential.



(1). 3D view of the STARVOC



(2). CFD-DEM simulation

Using computational fluid dynamics and the discrete element model (CFD-DEM) has gained popularity in research on particulate systems due to its ability to provide detailed information on particle scale, such as particle trajectories and the forces acting on an individual particle. With this method, transient data of individual particles, such as their position, velocity, and forces, can be obtained and analysed, which is essential to unravel the hydrodynamic fundamentals in the unit and to improve the design.

Program

- Literature study on state-of-the-art CFD-DEM.
- Gain acquaintance with the CFD package ANSYS Fluent and simulation on rotating reactor.
- Conduct CFD-DEM simulations, validated with experimental data.
- Simulation results processing and analysis.

1. Gonzalez-Quiroga, Arturo, et al. Powder Technology 354 (2019): 410-422..
2. Gonzalez-Quiroga, Arturo, et al. Chemical Engineering and Processing-Process Intensification 169 (2021): 108638.

GHENT UNIVERSITY

LCT DRIVING CHEMICAL TECHNOLOGY