Simulation of gas-liquid bubble column contactor using OpenFOAM

Kurucheti Rishabh Kumar

Department of Aerospace Engineering, Amrita Vishwa Vidyapeetham, Coimbatore, Tamil Nadu, India-641112

Guide: Dr Naren PR

Dean, Capacity Building & Digital Transformation Professor, Chemical Engineering School of Chemical & Biotechnology (SCBT)
SASTRA Deemed to be University, Thanjavur, Tamil Nadu, India - 613 401

Mentor: Mr Manjil Sitoula

FOSSEE IIT Bombay

Abstract

The objective of the present study is to numerically Simulate the gas-liquid bubble column contactor in open source CFD package OpenFOAM. This report will help the reader to understand the step-by-step procedure involved in simulating bubble column contactor and also explain the details regarding the boundary conditions, solver settings etc. In this simulation one can understand how two different types of fluid interact in multiphase flow. Air is being injected at superficial gas velocity into the column filled with water upto certain level, where the initial volume fraction distribution has been set in the "setFieldsdict" file. the transient hydrodynamics are resolved by solving the Euler–Euler multiphase governing equations. Geometry and mesh have been generated using 'blockMesh' utility available in OpenFOAM and flow has been simulated using "twoPhaseEulerFoam" solver. The air is injected through a sparger which is located at certain distance from one end of the column. The sparger location is constant throughout the simulation. Three different velocities have been considered in this case (0.2m/s, 0.4m/s and 0.6m/s). The results provide the information regarding the bubble rise behaviour, gas holdup, and flow regime development.

Reference: -

Nygren, Andreas. "Simulation of bubbly flow in a flat bubble column." (2014).