LEARNING COMPUTER FLUID DYNAMICS BY EXAMPLE: SIMULATING A FLUIDIZED BED UNIT AT THE LAB WITH OPENFOAM

CAÇO, ANA I.[1], MATA, T. M.[2] ,MARTINS, A A. [3] DA SILVA F., F. A.[4]
[1]*Chemistry Department, University of Aveiro / CICECO, icaco@ua.pt*
[2] *Faculty of Engineering University of Porto / LEPABE tmata@fe.up.pt*
[3] *Faculty of Engineering University of Porto / LEPABE amartins@fe.up.pt*
[4]*Chemistry Department, University of Aveiro / CICECO, fsilva@ua.pt*

Keywords: *Fluidized Bed, Sophomore Laboratory Experiments, Pressure drop, Bed void fraction*

Computer Fluid Dynamic (CFD) is an important topic in modern Chemical Engineering curriculum normally given in advanced research courses. However, only recently has been recognized the necesity to include this numerical paradigm earlier, in the core of regular pensum.[1] This is mainly due to the involved and detailed mathermatical models, usually partial diferential equations, which require sophisticate numerical methods for their solution, demanding time to domain and to develop solid skills by the students. Two formats that seems to be more efective to introduce this topic is by combination of tutorial training with laboratory experiments for comparing with[1]. Being OpenFOAM an opensource and robust option, ready available to tackle this kind of problems, it has been selected as CFD tool[2]. In the Chemical Engineering Integrated Master (MIEQ) course, from Chemistry Department of University Aveiro, we have Laboratory disciplines where it is possible to incorporate the CFD concept. One interesting laboratory practice, usually given at third year of MIEQ, named Laboratory of Chemical Engineering Science, is a work related with fluidized bed experiments [3]. Simultaneously, there are programming examples available as tutorial exercises in the OpenFOAM distribution, that could be used as a starting point to develop and solve this numerical problems. On of them, is the tutorial entitled A twophaseEulerFoam that seems to be a good starting point[4]. We proposed in this work, to adapt that tutorial to our experimental set-up, and perform simulations to obtain the pressure drop and average void fraction as function of fluid flow in the system, and compare with the CFD predictions. At the end of this work, we will propose a new experimental protocol assay amanable to be conferred to our students and share our experience while trying to setting up this challeging task.

Acknowledgements
This work was developed within the scope of the project CICECO-Aveiro Institute of Materials (Ref. FCT UID /CTM / 50011/2013), financed by national funds through the FCT/MEC and when appropriate co-financed by FEDER under the PT2020 Partnership Agreement.

References

[1] D. Adair, Z. Bakenov, e M. Jaeger, «Building on a traditional chemical engineering curriculum using computational fluid dynamics», *Educ. Chem. Eng.*, vol. 9, n. 4, pp. e85–e93, Dez. 2014.
[2] T. Marić, J. Höpken, e K. Mooney, *The OpenFOAM technology primer*. Duisburg: Sourceflux, 2014.
[3] «Laboratórios de Ciências de Engenharia Química. Guia de Trabalhos Práticos para Licenciatura em Engenharia Química.», Chemistry Department, University of Aveiro, Aveiro, 2016.
[4] S. Busch, «A twophaseEulerFoam tutorial», Chalmers University of Technology, Sweden, Tutorial, 2015.