STRUCTURED AND UNSTRUCTURED GRID VALIDATION OF A BUBBLE COLUMN REACTOR CFD MULTIPHASE MODEL BY ANSYS® WORKBENCH V10.0.

M. Martínez[*], R. Miró[†], S.C. Cardona[††], J. Navarro-Laboulais[††], S. Chiva[†††]

[*,†] Polytechnic University of Valencia (UPV)
Camí de Vera s/n, 46022 Valencia, Spain
e-mail: momarlia@iqn.upv.es, rmiro@iqn.upv.es

[††] Polytechnic University of Valencia (UPV-EPSA)
Pl. Ferrandiz y Carbonell s/n, 03801 Alcoi (Alicante), Spain
e-mail: scardona@iqn.upv.es, jnavarla@iqn.upv.es

[†††] Universitat Jaume I
Campus Riu Sec, 12071 Castelló de la Plana, Spain
e-mail: schiva@emc.uji.es

ABSTRACT

Bubble column reactors are widely used in industry as gas-liquid (G-L) reactors. Hydrodynamic characterization of vertical bubble column reactors is required to improve de global process performance of bubble column reactors [1]. In view of this, conductivity and pressure measurements have been made with tap-water/air phases in a vertical bubble column reactor with 1.535 m height and 0.19 m inner diameter. The gas distributor located at the bottom of the reactor consists of 127 x 0.4 mm diameter glass capillaries ensuring a uniform distribution of air in the bubble column. In addition, the process have been modeled and simulated by the commercial fluid dynamics computational code (CFD) ANSYS® CFX v10.0.

A three-dimensional (3D) transient model has been developed to simulate the local hydrodynamics of a gas-liquid two phase bubble column using the computational fluid dynamic method. Eulerian-Eulerian approach has been employed to simulate the multiphase process.

The generic CFD code ANSYS® CFX v10.0 [2] has been employed as a platform for two-fluid flow computation. Two grid types have been compared, structured and unstructured. An unstructured tetrahedral grid model with radial symmetry has been assumed and numerical simulations have been performed on a 60^{o} radial sector. Unstructured grid has been developed by ANSYS® CFX-Mesh module. A structured hexahedral model to the complete bubble column reactor has been calculated. The module ANSYS® ICEM CFD has been employed for the structured grid generation.

The CFD code has been executed in a cluster for scientific computational calculations. To save computing cost, a time-stepping strategy has been used. The local time-averaged gas hold ups, pressure and axial liquid velocities have been compared with experimental data obtained in the laboratory under varied air flow rate conditions [3]. Data results of structured and unstructured grid simulations have been compared to evaluate the best grid type for two-phase flow application of the bubble column reactor.

References

[1] J.B. Joshi, Computational flow modeling and design of bubble column reactors. *Chemical Engineering Science* **56**, pp. 5893–5933 (2001).

[2] CFX-10 User Manual. *ANSYS® CFX* (2005).

[3] S. Degaleesan, M. Dudukovik and Y. Pan, Experimental study of gas induced liquid-flows structures in bubble columns. *AICHE Journal* **47**, pp. 1913-1931 (2001).