APPLICATION OF CFD TECHNIQUES ON A DISTILLATION SIEVE TRAY FOR HYDRODYNAMIC PREDICTIONS

G. H. Justi; J. A. S. Gonçalves
Department of Chemical Engineering, Federal University of São Carlos
Via Washington Luiz, Km. 235, 13565-905, São Carlos, SP, Brazil
Phone: +55 (16) 3351-8045, Fax: +55 (16) 3351-8266, e-mail: jasgon@ufscar.br

ABSTRACT

A Computational Fluid Dynamics (CFD) model was developed to describe the hydrodynamics on a distillation sieve tray. We proposed a two-fluid, 3-D and steady state model to describe the gas and liquid phases, using the continuity and momentum conservation equations for an air/water system at 1 atm. An Eulerian-Eulerian framework and the standard k-ε turbulence model (only for the liquid phase) were used. The simulated tray had a diameter of 1.22 m. The simulation domain included the downcomer region. The model predicted the pressure and velocity fields, clear liquid height, froth height, volume fraction of each phase and recirculation zones for several combinations of gas and liquid flow rates. The results were compared with the experimental data of Solari and Bell (1986). A numerical unstructured mesh was constructed in a commercial package CFD software and the simulation was conducted using in an Intel®Corei7 Quad 2.97 GHz. The CFD model results show a chaotic hydrodynamics and liquid recirculation zones on the tray. The clear liquid height determined from the simulations presented a behavior close to the data. It is possible to conclude that CFD is an important tool in the design and optimization of sieve trays and also it can be used as a "virtual experiment" to simulate the fluid dynamics behavior on trays under several operating conditions.

Supported by Agência Nacional do Petróleo (ANP)

KEYWORDS

CFD; Distillation; Sieve tray; Clear liquid height; Froth height.