Flow Distributor Optimization in a Tubular Reactor

J. O. R. Lizarazo[1], L. A. S. Mejía[1], M. I. J. Gutierrez[1], J. A. P. Avella[1]

[1]Grupo de Investigaciones en Minerales, Biohidrometalurgia y Ambiente - GIMBA, Universidad Industrial de Santander, Bucaramanga, Santander, Colombia

Abstract

In a tubular reactor is the primary disturbing variable fluid flow is its geometry [1], ie, the load losses that occur in the fluid are caused mainly by the edge effects of that geometry and the compartment input and output flow rates of the reactor. It is also necessary to ensure good distribution of the liquid in order to improve mass transfer. [2-3]

In this work, the geometric of the entrance-exit distributors and their effect on flow characteristics was evaluated in a tubular reactor [Figure 1]. The CFD Module of the COMSOL Multiphysics® software was used as a tool to study the dynamic flow behavior of the fluid within the reactor.

The simulation of the fluid velocity field for different Reynolds numbers enabled the identification of recirculation zones and stagnant areas inside the reactor [Figure 1]. The modifications in the geometry of inlet and outlet fluid distributors provided important information to improve the hydrodynamic and minimize the formation of stagnant zones within the reactor.

From these results a new compartment was designed and the COMSOL Multiphysics® software was again employed to predict its performance.

Reference

[1] F. Gutiérrez Martín et al. Ingeniería de la reacción química. Cap 1, 3-8(2008).
[2] Á. Frías-Ferrer et al. Chem. Eng. J. 169 270–281(2011).
[3] J. Wang, et al. Electrochim. Acta, 173, 698–704(2015).

Figures used in the abstract

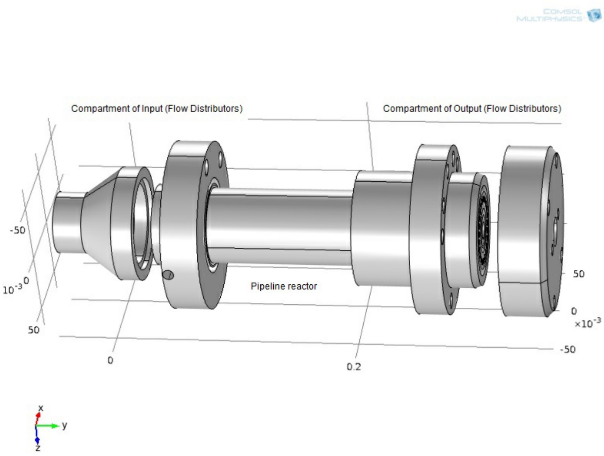

Figure 1: Tubular reactor modeled