World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:12, No:09, 2018

Exploration of Cone Foam Breaker Behavior Using Computational Fluid Dynamic

Authors : G. St-Pierre-Lemieux, E. Askari Mahvelati, D. Groleau, P. Proulx

Abstract : Mathematical modeling has become an important tool for the study of foam behavior. Computational Fluid Dynamic (CFD) can be used to investigate the behavior of foam around foam breakers to better understand the mechanisms leading to the 'destruction' of foam. The focus of this investigation was the simple cone foam breaker, whose performance has been identified in numerous studies. While the optimal pumping angle is known from the literature, the contribution of pressure drop, shearing, and centrifugal forces to the foam syneresis are subject to speculation. This work provides a screening of those factors against changes in the cone angle and foam rheology. The CFD simulation was made with the open source OpenFOAM toolkits on a full three-dimensional model discretized using hexahedral cells. The geometry was generated using a python script then meshed with blockMesh. The OpenFOAM Volume Of Fluid (VOF) method was used (interFOAM) to obtain a detailed description of the interfacial forces, and the model k-omega SST was used to calculate the turbulence fields. The cone configuration allows the use of a rotating wall boundary condition. In each case, a pair of immiscible fluids, foam/air or water/air was used. The foam was modeled as a shear thinning (Herschel-Buckley) fluid. The results were compared to our measurements and to results found in the literature, first by computing the pumping rate of the cone, and second by the liquid break-up at the exit of the cone. A 3D printed version of the cones submerged in foam (shaving cream or soap solution) and water, at speeds varying between 400 RPM and 1500 RPM, was also used to validate the modeling results by calculating the torque exerted on the shaft. While most of the literature is focusing on cone behavior using Newtonian fluids, this works explore its behavior in shear thinning fluid which better reflects foam apparent rheology. Those simulations bring new light on the cone behavior within the foam and allow the computation of shearing, pressure, and velocity of the fluid, enabling to better evaluate the efficiency of the cones as foam breakers. This study contributes to clarify the mechanisms behind foam breaker performances, at least in part, using modern CFD techniques.

Keywords : bioreactor, CFD, foam breaker, foam mitigation, OpenFOAM
Conference Title : ICCFDM 2018 : International Conference on Computational Fluid Dynamics and Mechanics
Conference Location : Tokyo, Japan
Conference Dates : September 10-11, 2018