عنوان مقاله:

Hydrodynamic Study in a Cone Bottom Stirred Tank Using Computational Fluid Dynamics

محل انتشار:

دوماهنامه مکانیک سیالات کاربردی, دوره 16, شماره 10 (سال: 1402)

تعداد صفحات اصل مقاله: 17

نویسندگان:

L. F. Cardona - *Pulp and Paper Research Group, Faculty of Chemical Engineering, Universidad Pontificia Bolivariana, Medellín ۵۶۰۰۶, Colombia*

J. E. Arismendy - *Pulp and Paper Research Group, Faculty of Chemical Engineering, Universidad Pontificia Bolivariana, Medellín ۵۶۰۰۶, Colombia*

G. C. Quintana - *Pulp and Paper Research Group, Faculty of Chemical Engineering, Universidad Pontificia Bolivariana, Medellín ۵۶۰۰۶, Colombia*

H. H. Alzate - *Pulp and Paper Research Group, Faculty of Chemical Engineering, Universidad Pontificia Bolivariana, Medellín ۵۶۰۰۶, Colombia*

خلاصه مقاله:

Stirred tanks are often used in industrial applications to store and process liquids and solids. However, these systems have become an increasing challenge to improve and optimize these processes. Computational Fluids Dynamics (CFD) simulation predicts complex phenomena as hydrodynamics system performance. An optimal solution is found using an effective mesh scheme and selecting appropriate boundary conditions. This work aims to validate and describe the distribution velocities inside the tank using a rigorous turbulence model. Stirred tank with a diameter of ۲۷ cm and an oval cone tip using a Rushton impeller (radial impeller) and a ۴-blade impeller inclined at ۴۵° (axial impeller) are performed. For both cases, hydrodynamics in the bottom tank is analyzed. In addition, the power and the pumping numbers for each impeller are studied. The overall results show that at the tip of the oval cone, the asymmetry in the mesh is improved, and the divergence in the solution is avoided. Also, the cone designer increased the turbulent kinetic energy, which can enhance the mixture process. A decrease in power impeller is shown when the axial type is applied at low Reynolds numbers; however, when the cone is introduced inside the tank and a radial impeller type is used, the impeller power values are increased. The overall results of CFD simulation are compared to experimental data and provide similar values with an absolute deviation below ۴.۴۶ %.

کلمات کلیدی:

Turbulence models, Impellers, Power number, Pumping number, Computational fluid dynamics, Turbulent kinetic energy

لینک ثابت مقاله در پایگاه سیویلیکا: