World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:16, No:12, 2022

Using Computational Fluid Dynamics (CFD) Modeling to Predict the Impact of Nuclear Reactor Mixed Tank Flows Using the Momentum Equation

Authors : Joseph Amponsah

Abstract : This research proposes an equation to predict and determine the momentum source equation term after factoring in the radial friction between the fluid and the blades and the impeller's propulsive power. This research aims to look at how CFD software can be used to predict the effect of flows in nuclear reactor stirred tanks through a momentum source equation and the concentration distribution of tracers that have been introduced in reactor tanks. The estimated findings, including the dimensionless concentration curves, power, and pumping numbers, dimensionless velocity profiles, and mixing times 4, were contrasted with results from tests in stirred containers. The investigation was carried out in Part I for vessels that were agitated by one impeller on a central shaft. The two types of impellers employed were an ordinary Rushton turbine and a 6-bladed 45° pitched blade turbine. The simulations made use of numerous reference frame techniques and the common k-e turbulence model. The impact of the grid type was also examined; unstructured, structured, and unique user-defined grids were looked at. The CFD model was used to simulate the flow field within the Rushton turbine nuclear reactor stirred tank. This method was validated using experimental data that were available close to the impeller tip and in the bulk area. Additionally, analyses of the computational efficiency and time using MRF and SM were done.

Keywords : Ansys fluent, momentum equation, CFD, prediction
Conference Title : ICCFDA 2022 : International Conference on Computational Fluid Dynamics and Applications
Conference Location : Paris, France
Conference Dates : December 29-30, 2022