World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:9, No:08, 2015

Numerical Simulation of Three-Dimensional Cavitating Turbulent Flow in Francis Turbines with ANSYS

Authors : Raza Abdulla Saeed

Abstract : In this study, the three-dimensional cavitating turbulent flow in a complete Francis turbine is simulated using mixture model for cavity/liquid two-phase flows. Numerical analysis is carried out using ANSYS CFX software release 12, and standard k-ε turbulence model is adopted for this analysis. The computational fluid domain consist of spiral casing, stay vanes, guide vanes, runner and draft tube. The computational domain is discretized with a three-dimensional mesh system of unstructured tetrahedron mesh. The finite volume method (FVM) is used to solve the governing equations of the mixture model. Results of cavitation on the runner's blades under three different boundary conditions are presented and discussed. From the numerical results it has been found that the numerical method was successfully applied to simulate the cavitating two-phase turbulent flow through a Francis turbine, and also cavitation is clearly predicted in the form of water vapor formation inside the turbine. By comparison the numerical prediction results with a real runner; it's shown that the region of higher volume fraction obtained by simulation is consistent with the region of runner cavitation damage.

Keywords : computational fluid dynamics, hydraulic francis turbine, numerical simulation, two-phase mixture cavitation model
Conference Title : ICCM 2015 : International Conference on Computational Mechanics
Conference Location : Venice, Italy
Conference Dates : August 13-14, 2015