Abstract

Ashraf I. Sharara

A Numerical Investigation of the Incompressible Flow through a Butterfly Valve using CFD

Abstract—The results of a numerical investigation of a butterfly valve under incompressible flow conditions are reported. The commercial CFD code Fluent was used for the study, and the standard K-? turbulence model was Selected. The simulations provided insight into the flow field characteristics of the valve. The valve performance factors such as the flow coefficient and hydrodynamic flow coefficient were computed for different valve opening angles. The results were compared with experimental data and good agreement was found. The results demonstrated the importance of CFD as a valuable tool in visualizing the complex structures in the flow field of the valve. The pressure across the valve and the valve flow coefficient were found to be directly dependent on the opening angle of the valve disk. The hydrodynamic torque coefficient curve was computed at different valve opening angles, and the angle at which maximum torque coefficient occurs was identified.