Numerical Study of a Butterfly Valve for Vibration Analysis and Reduction

Authors : Malik I. Al-Amayreh, Mohammad I. Kilani, Ahmed S. Al-Salaymeh

Abstract : This works presents a Computational Fluid Dynamics (CFD) simulation of a butterfly valve used to control the flow of combustible gas mixture in an industrial process setting. The work uses CFD simulation to analyze the flow characteristics in the vicinity of the valve, including the velocity distributions, streamlines and path lines. Frequency spectrum of the pressure pulsations downstream the valves, and the vortex shedding allow predicting the torque fluctuations acting on the valve shaft and the possibility of generating mechanical vibration and resonance. These fluctuations are due to aerodynamic torque resulting from fluid turbulence and vortex shedding in the valve vicinity. The valve analyzed is located in a pipeline between two opposing 90o elbows, which exposes the valve and the surrounding structure to the turbulence generated upstream and downstream the elbows at either end of the pipe. CFD simulations show that the best location for the valve from a vibration point of view is in the middle of the pipe joining the elbows.

Keywords : butterfly valve vibration analysis, computational fluid dynamics, fluid flow circuit design, fluctuation

Conference Title : ICMAME 2014 : International Conference on Mechanical, Aeronautical and Manufacturing Engineering
Conference Location : Sydney, Australia
Conference Dates : December 15-16, 2014