Numerical Study of a Butterfly Valve for Vibration Analysis and Reduction

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 pressure distributions and 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 90^o 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.

ICMAME 2014: XII International Conference on Mechanical,
Aeronautical and Manufacturing Engineering to be held in Sydney

M. Al-Amayreh is with the Mechanical Engineering Department at the AlZaytoonah. University of Jordan, P.O.BOX 130, 11733 AMMAN, Jordan (phone: 00962-779643854, e-mail: malik.amayreh@jeamail.org.jo).

M. Kilani, is with Department of Mechatronics at the university of Jordan, 11942 Amman, Jordan (e-mail: mkilani@ju.edu.jo).

A. Al-Salaymeh is with the Mechanical Engineering Department at the University of Jordan, 11942 Amman, Jordan (e-mail: salaymeh@ju.edu.jo).