GLOBE VALVE IN SINGLE PHASE FLOW: EXPERIMENTAL AND CFD ANALYSIS

Rémi SALANON[a], Loïc ANCIAN[a], Najib FAZAIL[b], Claude SOUPRAYEN[c]

[a] *Vibratec, 28 chemin du petit bois, Ecully Cedex BP36-69131, France*
[b] *Vibratec Asia Pacific, Jalan Bangsar Utama 1, Bangsar, 59000 Kuala Lumpur, Malaysia*
[c] *Fluidyn, 84 Rue Charles Michels, 93200 Saint-Denis, France*

Abstract
The potential of flow-induced vibration (FIV) to cause catastrophic failure of engineering systems and unacceptable levels of environmental and occupational noise has motivated significant effort to understand and mitigate the problem in the interests of human safety. Valves can regulate, direct, or control the flow of a fluid by opening, closing, or partially obstructing various passageways. Inside the valve, mechanical loads are generated due to turbulence. These mechanical loads can induce excessive vibration in the flow lines and may cause fatigue failure in the piping, support structures, and welds. In order to have a better understanding of phenomena occurring in globe valves, it was required to combine experimental and numerical approaches. The present paper provides such numerical analysis with computational fluid dynamics (CFD) solutions for the flow, deployed within a qualification step, for the re-production of data from the experiments, and for extended flow regimes. Measurements were conducted to determine the pressure fluctuations induced by the turbulent loads generated by the globe valve. A test loop has been designed in order to investigate the water flow behavior in a piping system passing through the globe valve. Nine pressure sensors were installed in order to measure pressure fluctuations in each potential area of interest. Several flow rates and valve openings were tested. Dimensionless power spectral density (PSD) pressure fluctuations were post-processed to be compared later with CFD computation results. To understand the behavior of the flow, steady simulations of the flow have been performed first. To further analyze and produce quantitative numerical data for comparisons with the experiments, the computational fluid dynamics – large eddy simulation (CFD-LES) simulations have been carried out. The analysis of pressure fluctuations at sensor positions and comparisons with the experimental data provide a quantitative assessment of the turbulence intensities and loads on the inner faces of the structure. Finally, a larger-scale industrial valve, out of the scope of the capabilities of the experimental bench, is studied for a potential extension of the results and conclusions for a design strategy. Two scaling protocols have been selected in order to test the numerical feasibility of high-resolution simulations on industrial-scale equipment and the relevance of the proposed re-scaling laws on different dimensions.