Flow dynamics in a pipe with a sudden change in diameter

Ivar Annus, Katrin Kaur, Anatoli Vassiljev, Janek Laanearu and Märt Šanin
Department of Mechanics
Tallinn University of Technology

The calibration of a numerical model of an existing Water Distribution System (WDS) that contains old pipes is usually based on the estimation of the roughness of the pipes. Due to the lack of information, nominal pipe diameters are generally used in the model development and the change in pipe diameter due to pipe wall build-up is compensated by adjusting the roughness value [1]. This can lead to quite unrealistic roughness values. Information on pipe diameters and flow dynamics is very important for estimating the propagation rate of the contaminated zones in case of chemical or biological threats [2]. Information about the real flow velocities and pressure losses is essential.

To address the needs, an experimental study was conducted to analyse the flow dynamics in a pipe with a sudden change in diameter. A pipeline apparatus at the Laboratory of Fluid Mechanics, Tallinn University of Technology was used to investigate the velocity and pressure changes at different flow rates. The experimental apparatus depicted in Figure 1 consist of a tank at the upstream end, a horizontal transparent pipe of total length L = 18.8 m with the internal diameter D = 80 mm and a control valve at the downstream end. The 2 m long test section (Figure 1) consists of two short pipes in both ends with internal diameter D = 80 mm, two longer pipes with internal diameter D = 46.4 mm and a 0.23 m long transparent section with internal diameter D = 80 mm in the middle. Changes in velocity distribution in case of a sudden change in diameter were measured in the middle section using two-dimensional PIV (Particle Image Velocimetry). In addition pressures were measured at 3 different locations over the pipe length and flow rate was measured with an electromagnetic flowmeter installed at the downstream end of the pipe.

The experimental results were used to validate two different CFD (Computational Fluid Dynamics) simulation software. 3D analyses were conducted in a commercial software ANSYS Fluent and an open source software OpenFOAM. In ANSYS Fluent K-epsilon turbulence model was used in the simulations. Wall functions and turbulence parameters were validated using measured data. In OpenFOAM a standard solver simpleFoam was used. It is a steady-state solver for incompressible flows with turbulence modelling and is based on K-epsilon model. Measured and calculated flow rates and velocity distributions were compared.

Experimental investigations revealed that in a pipe section with a sudden increase of diameter at the inlet and sudden decrease of diameter at the outlet the velocity distribution is dependent on the flow rate. The pipe diameter where velocity is positive (effective diameter) is smaller than nominal and velocity much higher than expected. Analysis of the axial velocity distributions at different distances from the inlet showed that in the core region the velocity profile is uniform and in the near-wall region the velocity is close to zero or negative. CFD results are qualitatively comparable with experimental results but need to be calibrated to coincide with measurements in the near wall region. The sensitivity of different CFD input parameters is analysed to find better qualitative and quantitative correlation between measurements and modelled results.

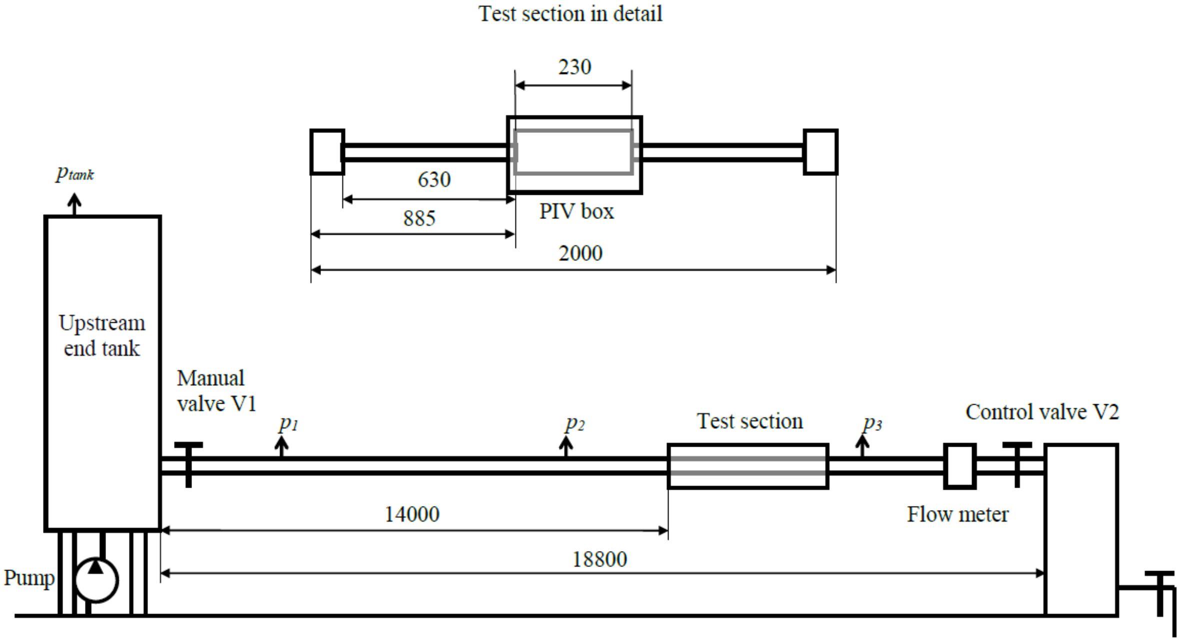

Figure 1 – Schematic of the test rig

References

[1] K. E. Lansey, W. El-Shorbagy, I. Ahmed, J. Araujo, C. T. Haan, Calibration assessment and data collection for water distribution networks, Journal of Hydraulic Engineering, 127(4), 2001, pp. 270-279.
[2] A. Vassiljev, T. Koppel, Estimation of real-time demands on the basis of pressure measurements by different optimization methods, Advances in Engineering Software, 80, 2015, pp. 67-71.