Numerical simulation of flow metering system for liquefied natural gas with nozzle shape optimization

J. Sluse[1], R. Maury[2],
J. Gersl[1], A. Strzelecki[2]

[1]Czech Metrology Institute, Okruzni 31, 638 00 Brno, Czech Republic
[2]CESAME Exadebit, Poitiers, France
E-mail (corresponding author): jsluse@cmi.cz

Increasing usage of liquefied natural gas (LNG) hand in hand with the decreasing price makes this „green“ fuel accessible. Higher consumption of LNG opened gate for development of new facility for measuring of flow rate with lower uncertainty. The new equipment for measuring the LNG flow rate is using a Laser Doppler Velocimetry technique. The principle of the equipment is based on the velocity measurement in one point behind convergence nozzle and then solving the flow rate. It is necessary to create flat velocity profile behind the nozzle throat (like piston velocity profile) to reduce the shear region influence on the mass flow rate calculation.

This problem is solved by numerical simulation (CFD) using open-source program OpenFOAM. The numerical simulation study is divided into following parts:

- The first part is dedicated to the flow through a long straight pipe and a standard Venturi tube (ISO 5167-4). Mentioned cases are simplified version of the LDV package system. In these cases some analytical and experimental results are known [1], [2]. Mesh convergence and choice of a turbulence model are checked for these cases. The numerical validation is realized by comparing analytical, experimental and numerical aerodynamic behaviour of the flow. Furthermore, the outlet solution obtained for the straight pipe case is used as inlet boundary condition for the LDV package system.

- The aim of the next part of this article is to use the knowledge acquired from the previous calculations for the LDV package system (meshes, turbulence model and boundary conditions). Then the numerical simulation of the LDV package is compared with experimental data from air, liquid nitrogen and LNG flow.

- Last part includes a nozzle shape optimization to reduce the shear region impact on the mass flow rate calculation (velocity profile as flat as possible downstream the nozzle throat). Final shape optimization will be realized by making velocity profile comparison between several 3D models.

Bibliography

[1] ZAGAROLA MARK V., ALEXANDER J. SMITS. Mean-flow scaling of turbulent pipe flow. Journal of Fluid Mechanics. 373, 33-79. DOI: 10.1017/S0022112098002419. ISSN 00221120.
[2] WOSNIK MARTIN, LUCIANO CASTILLO, WILLIAM K. GEORGE. A theory for turbulent pipe and channel flows. Journal of Fluid Mechanics. 421, 115-145. DOI: 10.1017/S0022112000001385. ISSN 00221120.