Numerical and experimental investigation of the cavitating flow within Venturi tube

Jiří Kozák, Victor. Kaplan dpt. of Fluid Eng., Brno University of Technology, Czech Republic

Pavel Rudolf, Victor Kaplan dpt. Of Fluid Eng., Brno University of Technology, Czech Republic

Martin Hudec, Victor Kaplan dpt. Of Fluid Eng., Brno University of Technology, Czech Republic

Matěj Forman, ESI Group, Czech Republic

Long Abstract

Introduction

Hydrodynamic cavitation represents complex physical phenomenon undesirably affecting operation as well as lifespan of many hydraulic machines from small valves to the large hydro plants. On the other hand, the same phenomenon and its concomitants such as pressure pulsations can be exploited in many positive ways. One of the positive and perspective way of the cavitation utilization is reduction of the organisms such as cyanobacteria within large bulks of water. [1]

This application was the initial impulse of the cavitation dynamics investigation within the axisymmetric Venturi tube in the Victor Kaplan dpt. Of Fluid Engineering. The investigation has been focused on the study of the flow within the Venturi tube with a predominantly axial inflow in the first step. [2],[3]

In the second step of the investigation, the upstream mounted swirl generator has been exploited to introduce swirl to the flow. The additional swirl has been highly affecting the cavitation within the Venturi tube. Beside the change of the noise and pressure pulsations intensity, the significant change of the cavitation patterns has been observed in the whole range of the investigated cavitation regimes. Although the fact that a significant amount of experimental data has been acquired, the numerical analysis of the problem, which is main scope of the contribution, can provide additional information. It can be crucial for a better understanding of the phenomenon, mechanisms of the observed cavitation regimes, and above that for the correct interpretation of the experimental data.

1. Methods

The experimental measurement has been done using cavitation rig of the Victor Kaplan dpt. of Fluid engineering. It should me mentioned, that thanks to the vacuum pump and compressor connected to the pressure tank of the cavitation rig, it was possible to investigate wide range of cavitation regimes for constant discharges. The discharges of 4, 5, 6 and 7 l/s were experimentally investigated for the configurations with and without the swirl generator.

Although the significant amount of experimental data has been acquired during the measurement (including pressure pulsation, high speed video, vibrations, acoustic emission and noise records), it has not been possible to carry out an analysis of the velocity field within the nozzle. Therefore, but not only for this purpose, numerical analysis of the cavitating flow has been done.

The numerical analysis was focused on the investigation of the cavitating flow affected by the presence of the swirl generator. For the purpose of the CFD computations, discharge of 6 l/s was chosen. Computational grids were created using the CfMesh v1.1.1 and its predominantly hexahedral cartesianMesh algorithm. [5] Complete section, including the swirl generator, was considered in the first step of numerical analysis. The influence of the computational grid was partially studied using two different settings of the grid generator. In case of the finer mesh the cells were significantly refined mainly in the region of the nozzle axis downstream the spike of the swirl generator, which is depicted in the figure 1.

In the second step of the numerical investigation, the swirl generator has been excluded to reduce size of the computational grids and consequently reduce time and computational demands of the simulations. In these simplified simulations, the time-averaged velocity field and fields of turbulent kinetic energy and turbulent dissipation rate obtained downstream the swirl generator during the simulation of the whole geometry were exploited as inlet boundary conditions. Numerical analysis has been done using the OpenFoam 1606+ and its InterPhaseChangeFoam solver. Based on the previous experiences, realizable k-ε model of turbulence and Kunz model of mass transfer have been exploited. [4], [6] Length of the time step was driven by the maximum value of the Courant number 0.2. This setting resulted to the approximate time-step size of 6e-7s in case of the finer mesh including the swirl generator. The comparison of the cavitating vortex captured during the experimental measurement and numerical simulation is depicted in the figure 2.

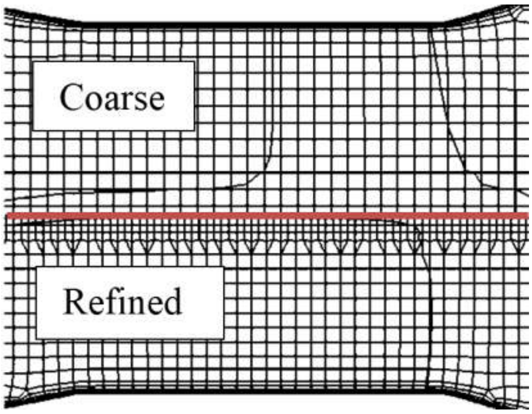

Figure 1 Comparison of the grid density in the region of the throat of the nozzle

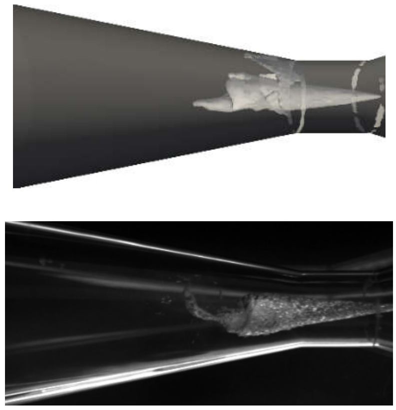

Figure 2 CFD results vs. captured cavitating vortex within the nozzle

Acknowledgement

Czech Science Foundation is gratefully acknowledged for support of this research under project No 16-18316S „Principles and mechanisms causing microorganism elimination by hydrodynamic cavitation “.

References

[1] D. Jančura, P. Mikula, B. Maršálek, P. Rudolf, F. Pochýly, Selective method for cyanobacterial bloom removal: hydraulic jet cavitation experience, *Aquaculture International*, 22 : 509-521, 2014.

[2] J. Kozák, P. Rudolf, D. Štefan, R. Huzlík, M. Hudec et al, Transition of cavitating flow to supercavitation within Venturi nozzle – hysteresis investigation, *EPJ Web of Conferences*, 2017.

[3] P. Rudolf, M. Hudec, M. Gríger, D. Štefan, Characterization of the cavitating flow in converging-diverging nozzle based on experimental investigations, *EPJ Web of Conferences*, 2015.

[4] J. Kozák, P. Rudolf, D. Štefan, M. Hudec, M. Gríger, Analysis Of Pressure Pulsations Of Cavitating Flow In Converging-Diverging Nozzle. *6th IAHR International Meeting of the Workgroup*,2015.

[5] F. Juretić, cfMesh User Guide v 1.1, 2015.

[6] Kunz, R.F., Boger, D.A., Stinebring, D.R., Chyczewski, Lindau. J.W., Gibeling, H.J., Venkateswaran, S., Govindan, T.R., A Preconditioned Implicit Method for Two-Phase Flows with Application to Cavitation Prediction, *Computers and Fluids*, 29(8):849-875, 2000.