6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, September 9-11, 2015, Ljubljana, Slovenia

ANALYSIS OF PRESSURE PULSATIONS OF CAVITATING FLOW IN CONVERGING-DIVERGING NOZZLE

Jiří KOZÁK*

Victor Kaplan Dept. of Fluid Engineering, Brno University of Technology, Czech Republic

Pavel RUDOLF

Victor Kaplan Dept. of Fluid Engineering, Brno University of Technology, Czech Republic

David ŠTEFAN

Victor Kaplan Dept. of Fluid Engineering, Brno University of Technology, Czech Republic

Martin HUDEC

Victor Kaplan Dept. of Fluid Engineering, Brno University of Technology, Czech Republic

Milan GRÍGER

Victor Kaplan Dept. of Fluid Engineering, Brno University of Technology, Czech Republic

ABSTRACT

Cavitating flow in converging-diverging nozzle (axisymmetric Venturi tube) is investigated. Characterization of the cavitating flow was done with high speed camera and the basic flow patterns were determined: incipient cavitation, partial cavitation, fully developed cavitation and supercavitation. Pressure transducers of tenzometric and piezoelectric types were used to determine amplitude-frequency characteristics of the pressure pulsations in particular regimes. Results were correlated with spectral properties obtained from high-speed camera video sequences and image processing analysis. Finally, attempt of unsteady cavitating flow CFD simulation is presented.

KEYWORDS

Cavitation, CFD, OpenFoam, Venturi tube, Vortex ring, FFT

1. INTRODUCTION

Cavitation is a complex physical phenomenon which occurs in liquid continuum when the static pressure drops below saturated vapour pressure. This pressure drop leads to creation of discontinuity filled by water vapour and undissolved gasses in the liquid continuum. The whole process of creation, existence and implosion of these cavities is accompanied with many phenomena (i.e. pressure fluctuations, acoustic emissions, surface erosion), which usually have undesirable impact on the operation of hydraulic machines. [1] On the other hand cavitation can be used in many positive ways, e. g. mitigation of microorganisms. These reasons lead to extensive research of this phenomenon in many branches of technical and medical engineering.

* *Corresponding author:* Victor Kaplan Dept. of Fluid Engineering, Brno University of technology, Technická 2, Brno, Czech Republic, email: jiri.kozak1@gmail.com

This paper is focused on the research of cavitation dynamics in the axisymmetric converging-diverging nozzle. The whole research was motivated by the effort to better understand this particular problem for the purpose of the water disinfection. [2]
The investigations of the cavitating flow in converging-diverging nozzle were done by many authors, on the other hand lots of published experiments were done using simplified geometry of the nozzle (2D plane C-D nozzle) and therefore these experiments were not capable to capture quite interesting phenomena of the vortex ring separation. [3]
It should be mentioned that the experimental research of the C-D nozzles integral characteristics and methodology of the loss coefficient evaluation of this experimental measurement was published by Rudolf et. al. [4]
Another investigation of the CFD analysis of cavitation flow downstream the axisymmetric Venturi tube were published by Kozubková et. al. [5]

2. Cavitation circuit

The experimental part of the work was carried out using the converging-diverging nozzle (Fig. 1) made of the block of cast plexiglass. Surfaces of the acrylic glass were polished for the purpose of the following high-speed video analysis. The edges of the C-D nozzle were made sharp, thus the location of the cavitation inception (i.e. location of the flow separation) could be located precisely.

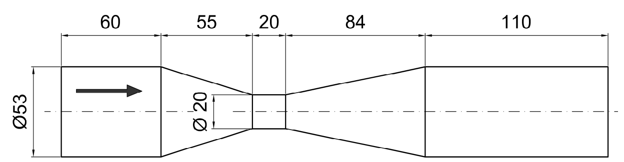


Fig. 1: Dimensions of the converging diverging nozzle

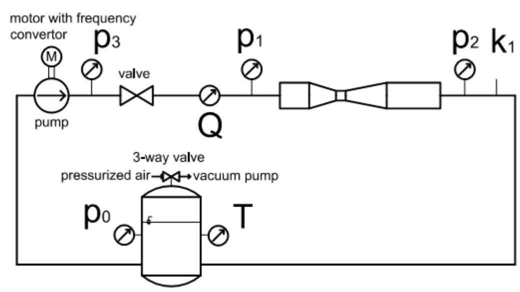


Fig. 2: Cavitation test rig

Cavitation rig (Fig. 2) allowed to modify the cavitation number by the change of the pump discharge or by adjusting of the pressure in the vessel which was equipped by the 3-way valve. Two different set-ups of the experiment were considered a) with the open pressure vessel (i.e. open to atmospheric pressure) and variable flow rate and b) with closed vessel with stable flow rate and variable static pressure level.
Discharge was measured using the induction flowmeter (Q). Static pressure were captured using the strain gauge pressure transducers (p_0, p_1, p_2, p_3), while for the more accurate capturing of the pressure fluctuations ultrafast piezoelectric pressure transducer (k_1) was employed. The temperature (resistance thermometer, T) as well as static pressure (p_3, strain gauge pressure transducers) were measured in the pressure vessel. Comprehensive characteristics of the utilized transducers are listed in Table 1.

Positions	**Manufacturer**	**Type**	**Range**	**Accuracy** (% of range)
p_0,p_1,p_2,p_3	BD Sensors	DMP 331	600 kPa, 250 kPa	± 0.25 %
k_1	Kistler	701A	250 bar	± 0.5 %
T	HIT	HSO-502 1A2L	0 – 50 °C	± 0.1 %
Q	ELA Brno	MQI99 SN	20 l/s	± 0.5 % *

Table 1: List of the transducers (0.5 % of the measured flow rate)*

Development of the cavitation in the nozzle was captured using the DSLR camera Nikon D300s and high-speed camcorder Baumer HXC 20 with the sampling frequency of 580 monochrome pictures per second with 0.5 Mpx resolution. Proper level of the illumination was provided by two halogen light sources with total output of 1000 W.
Unfortunately significant part of the measurements was done without the high speed piezoelectric Kistler pressure transducer, thus FFT of this operating point was not so precise compared with measurements utilizing the high speed pressure transducers.
Several pressure transducers were destroyed during the experiments due to the high amplitude of pressure fluctuations, which led to relocation of the pressure transducer downstream the nozzle and installation of the dampening rubber hose. Integration of this dampening element resolved the problem of the dangerous high pressure amplitudes on the one hand, on the other it had made impossible to compare pressure amplitudes.

3. Evaluation of the experimental measurements

State of the cavitating flow is usually described using the non-dimensional cavitation number σ. Despite widespread utilization of the cavitation number, its definition can differ significantly according to physical and geometrical properties of the investigated flow.
Cavitation number used throughout this paper is defined using the downstream pressure and velocity in the throat of the C-D nozzle:

$$\sigma = \frac{2 \cdot \left(p_2 - p_{vap}\right)}{\rho \cdot v_{throat}{}^2} \tag{1}$$

Dissipation of the energy was studied using the non-dimensional loss coefficient ξ expressed by the equation (2).

$$\xi = \frac{2 \cdot (p_1 - p_2)}{\rho \cdot v_{throat}{}^2} \tag{2}$$

Dynamics of the pressure pulsations was investigated using Fast Fourier Transformation (FFT). In case of high speed video analysis, the FFT input was value of pixel intensity, where the chosen pixel was always situated downstream the nozzle. Exact position differed, depending on the actual cavitation pattern. This method of the cavitating flow analysis using high-speed video record is well established and its accuracy has been documented for example by Sedlář et. al. in investigation of cavitation cloud shedding around NACA hydrofoil investigation. [6]

4. Numerical analysis

The computational mesh consisting 936 390 hexahedral cells of has been made using meshing software Gambit (Fig. 3). Thanks to the geometrical simplicity of the investigated case, the quality of the cell was guaranteed. The mesh has been finer in the near-wall region and in the diffuser part of the nozzle (e.g. in the assumed location of the cavitation, Fig. 4).

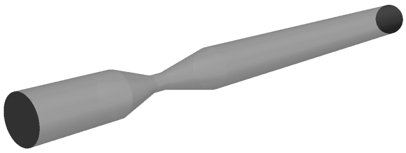


Fig. 3: Computational domain

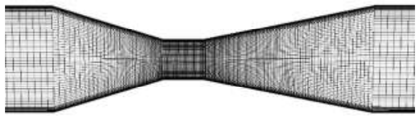


Fig. 4: Computational grid

The numerical analysis of the cavitating flow has been done using opensource CFD code OpenFoam 2.2.2. InterPhaseChange solver was utilized due to its capabilities to simulate cavitating flow considering mixture approach and Kunz model of cavitation has been chosen.[7]

Turbulent flow simulation relied on realizable k-ε model of turbulence, which proved to be suitable for similar class of problems. [8]
Constant value of velocity was assigned at inlet, while constant value of the absolute pressure was prescribed at the outlet of the domain.

5. Cavitation patterns

It should be mentioned that the transition between cavitation regimes was estimated only roughly using the change of the acoustic emission character accompanies transition between cavitation patterns.
First signs of the cavitation presence (i.e. cavitation inception) were detected when cavitation number dropped below the value between 0.84 and 0.93. Partial cavitation accompanied with severe pressure pulsations, vibrations and acoustic emissions has been observed until cavitation number reached value between 0.34 and 0.49, when fully developed cavitation was established. Finally, supercavitation with coherent water jet surrounded by the vapor volume appeared when cavitation number dropped below the value between 0.17 and 0.28.

6. Experimental measurement with constant pressure in the vessel

During this part of experimental measurement, the 3-way valve of the pressure vessel was fully opened, thus the variation of the cavitation number was caused only by the increase of the flow rate. (Fig. 6)
This experimental set – up was extended by the high-speed video analysis, where the video was treated using script written in Matlab. Input data for the FFT were consisted of pixel intensity value (the only property of the grayscale pixel). Therefore it is impossible to evaluate the amplitude of the pressure fluctuations. On the other hand, it is obvious that the pixel intensity is strongly tied with the presence and behavior of the cavitation cloud. Thus it could be assumed that video analysis is capable to provide relevant data related to the cavitation dynamics.
Cavitation dynamics was evaluated in range of cavitation numbers from 0.28 (supercavitation) to 0.8 (cavitation inception).
The evaluation in region of supercavitation was nearly impossible thanks to the fact that the pressure transducer was located in the region of the cavitation void. The problem was nearly the same in case of picture analysis, where it was not possible to find pixel with intensity variation related to the pressure pulsations. Comparison of the pressure record analysis and the analysis of the high-speed video is depicted in Fig. 5.

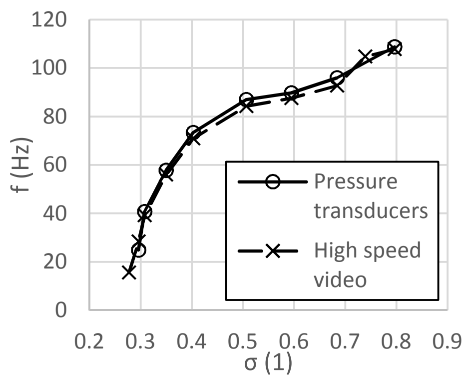


Fig. 5 Analysis of the pressure fluctuations: open vessel

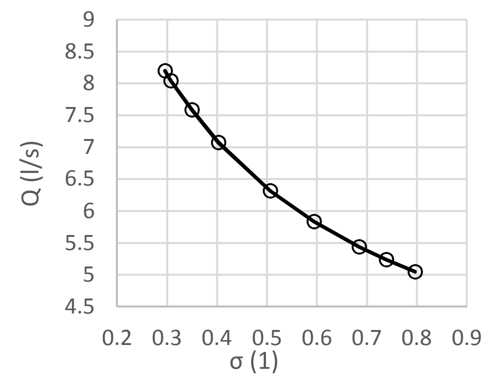


Fig. 6 Flow rate during the experiment: with open vessel

Results of both types of evaluation were in good agreement and the difference of the dominant frequency did not exceed 4% in any of the investigated operating points. Therefore it could be assumed that both of the methods are equivalent in the dominant frequency. The information of the actual flow rate corresponding to the individual cavitation numbers is depicted in the Fig. 6.

7. Experimental measurement considering constant flow rate

Experiments considering steady flow rate and variable pressures in the pressure vessel will be discussed in this part of the paper. Discharge of the pump ranged from 4 l/s to 10 l/s, thus in case of the lowest and highest flow rates the limit operational values of the pressure in the pressure vessel were reached (limits are imposed by the minimum pressure achieved by vacuum pump). Therefore flow-rate of 4 l/s was excluded from the discussion. This group of measurements was not accompanied by the high-speed video recording, except the discharge of 7 l/s, which will be described in detail in the following chapter. Results were obtained using high-speed piezoelectric pressure transducer in case of the flowrates of 7, 9 and 10 l/s, in the remaining flow rates, only the strain gauge pressure transducer was utilized. Strain gauge pressure transducer provided relevant results in case of higher cavitation numbers. The values of the pressure pulsations frequencies for varying cavitation number are depicted in the following chart.
Most of the curves depicted in Fig. 7 cover cavitation regimes from partial to fully developed cavitation. The only exceptions are flow rate of 6 l/s, which is extended to the cavitation inception and flow rate of 10 l/s covering the region of fully developed cavitation.
It could be seen that the slope of the curves is higher, the higher is actual flow rate. In the other words, in case of the same value of the cavitation number, higher value of the pressure fluctuations frequency could be expected in case of higher flow rate.

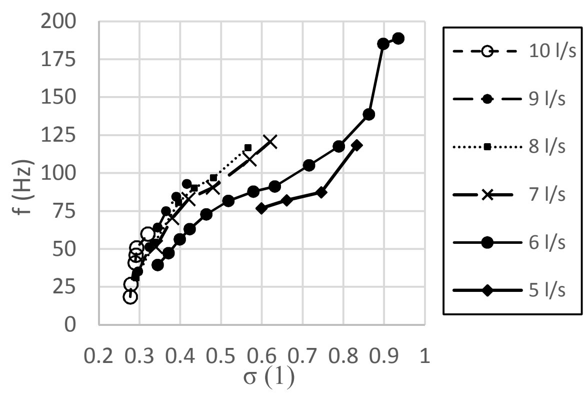


Fig. 7 Experimentally evaluated dominant frequencies in case of stable flowrate

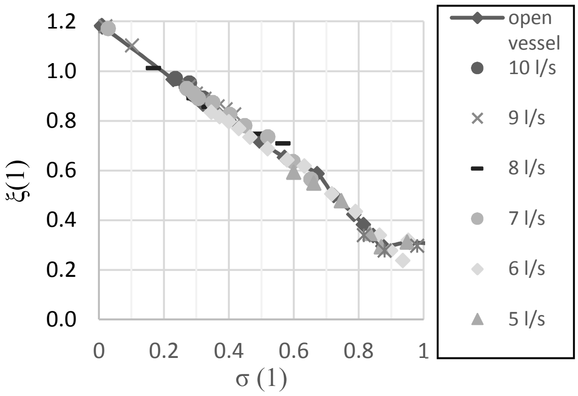


Fig. 8 Experimentally evaluated dominant frequencies in case of stable flowrate

The comprehensive information about methodology of the nozzle hydraulic loss can be found in [4] as it was mentioned earlier. However overall comparison of the loss coefficient for different flow rates is depicted in the Fig. 8.

8. Comparison of experimental and numerical results

The results of experimental measurement considering steady flow rate of 7 l/s with numerical analysis of the cavitating flow will be discussed in this chapter.
As well as in case of open vessel, the experimental measurement was extended by high-speed video recording, thus it is possible to compare all three methods of the cavitation assessment.
Partial cavitation, fully developed as well as supercavitation were investigated, where it was not possible to determine pressure pulsations during the supercavitation due to the reasons which were mentioned in previous chapter.

Cavitation pattern had changed its character from partial cavitation to fully developed cavitation when the cavitation number dropped below the value of 0.37. Further decrease of the pressure pulsations and the related undesirable effects occurred at cavitation number of 0.27 when the fully developed cavitation transformed to supercavitation.

The results of the cavitation dynamics analysis are depicted in the Fig. 9. It can be seen that the results of the pressure records and high speed analysis is in good agreement in the whole range of investigated operation regimes and the evaluated frequencies are nearly the same in case of both methods

The slope of the curves is more stable than in case of the results considering open vessel. From the point of multiphase numerical analysis view, the good agreement with experimental results could be seen in case of higher cavitation numbers (e. g. in the region of the partial cavitation onset).

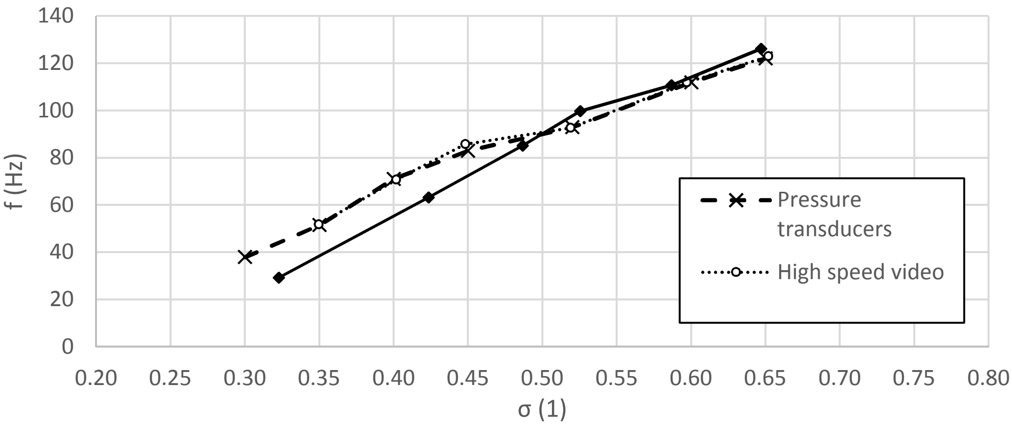
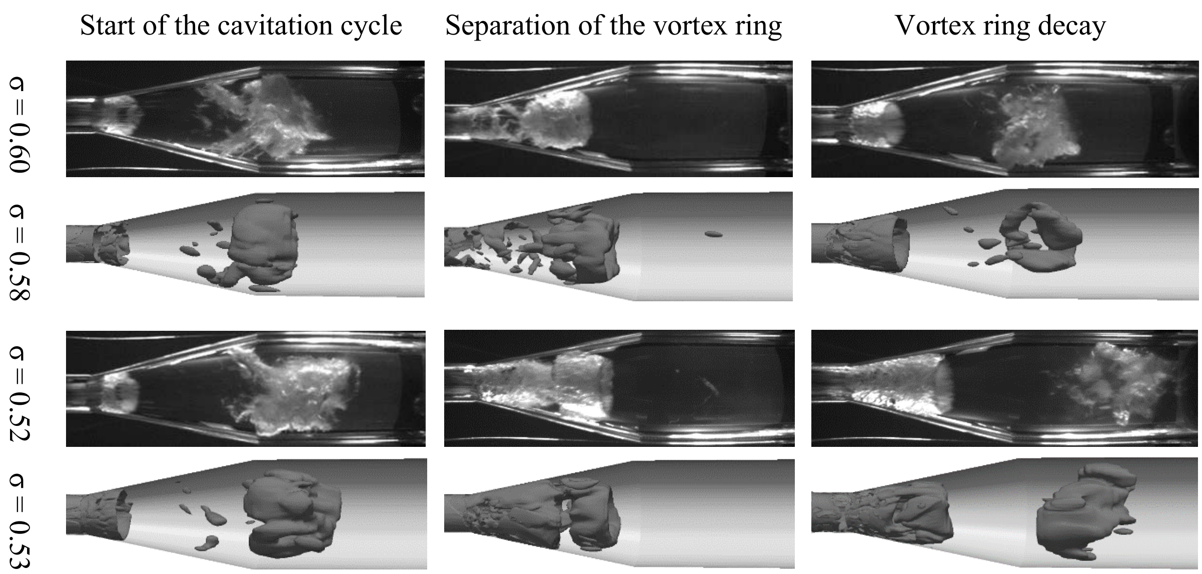


Fig. 9 Comparison of dominant frequencies for flow rate 7l/s

More significant difference could be found in the region of the transition from the partial to fully developed cavitation and in the region of fully developed cavitation itself where the numerically predicted frequency of the pressure pulsation is nearly twice lower compared to the experiments.

On the other hand OpenFoam with applied numerical set-up was capable to predict decrease of the frequency with the decreasing cavitation number and moreover it was capable to predict change of the cavitation patterns. Results of the numerical simulations and high speed video record are compared in the following sets of images (Fig. 10).



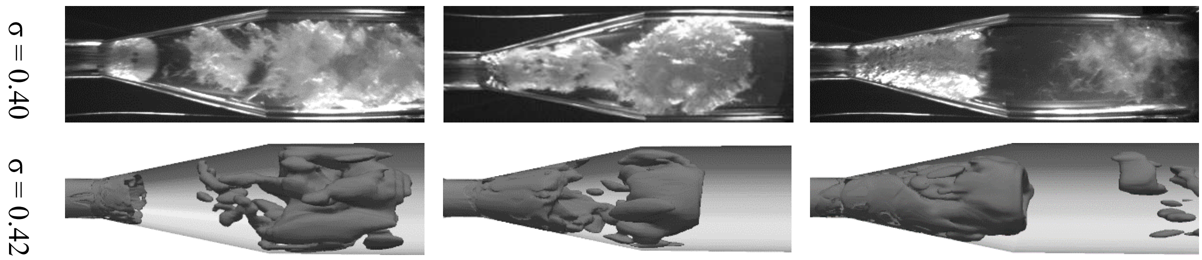


Fig. 10 Comparison of the numerical results with high speed video records.

Location of the vortex ring separation and its decay were well predicted in all of the presented operating points and even in the last operating point (σ = 0.42), where the frequency of the pressure pulsation is significantly lower than in reality (Fig. 9).
Attempt at supercavitation simulation was carried out. It should be emphasized that the downstream part of the computational grid was too short for this purpose. As a result of the inadequate length of the domain, the volume of cavitation was periodically collapsing during the simulation. On the contrary, the water jet behind the throat of the nozzle was stable during the experiments and the volume of cavitation was collapsing in the pipeline elbow several meters downstream the nozzle (Fig. 11).

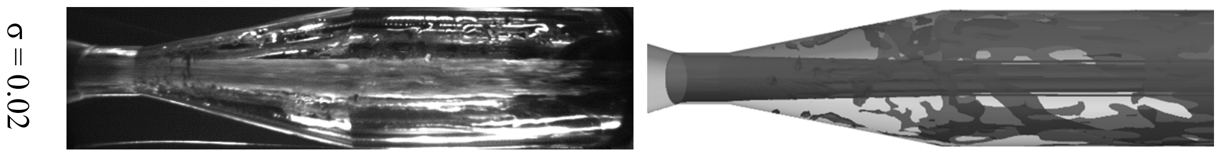


Fig. 11 Supercavitation regime, experiment (left,) numerical simulation with contour corresponding to 99 % of liquid water (right)

9. Conclusions

The comprehensive investigation of the cavitation flow downstream the C-D nozzle was presented. The dynamics of the cavitation flow was investigated using the experimental and numerical methods. It should be mentioned that the values of the pressure amplitudes were not compared due to necessity to change position of the pressure transducer and installation of the damping hose during the experimental part of the work.
The dominant frequencies of the pressure fluctuation obtained by high-speed video analysis were in correlation with the measured pressure fluctuations in case of the open vessel and in case of closed vessel and stable flow rate of 7 l/s.
Analysis of the pressure fluctuations during the experiments considering stable flow rate is depicting that the increase of the pressure fluctuations frequency with decreasing value of cavitation number is the steeper, the higher is the flow rate.
The results of numerical computations were satisfying in case of higher values of cavitation number where the frequencies of the pressure fluctuations were well-predicted In case of lower cavitation numbers, significant difference between numerical and experimental results were obtained. On the other hand the position of the cavitation ring separation and its decay was well predicted even in case of the lower values of the cavitation number.
Attempt to numerical computation of the supercavitation was carried out too. Despite the fact, that the computational domain was too short for the simulation of this cavitation regime, the results seems to be promising.

10. REFERENCES

[1] Brennen, Christopher E.: *Cavitation and bubble dynamics*. New York: Oxford University press. 1995.

[2] Jancula D., Mikula P., Marsalek B., et al.:*Selective method for cyanobacterial bloom removal: hydraulic jet cavitation experience* ,AQUACULTURE INTERNATIONAL Volume: 22.2014

[3] Decraix J., Goncalvès E.: *Investigation of three-dimensional effect on a cavitating Venturi flow.* International Journal of Heat and Fluid Flow 44. 2013.

[4] Rudolf P., Hudec M., Gríger M., Štefan D.: *Characterization of the cavitating flow in converging-diverging nozzle based on experimental investigation*. EPJ Web of conferences 67. 2014

[5] Kozubková M., Rautová J.: *Cavitation modelling of the flow in Laval nozzle.* 3rd IAHR International meeting of the Workgroup on Cavitation and Dynamic Problems on Hydraulic Machinery and systems. Brno. 2009

[6] Sedlář M., Komárek M., Rudolf P., Kozák J., Huzlík R.: Numerical and experimental research on unsteady cavitating flow around NACA 2412 hydrofoil. International Symposium of Cavitation and Multiphase Flow. 2014.

[7] Kunz, Robert F., Boger, David A., Stinebring, David R., Chyczewski, Thomas S., Gibeling, Howard J., Venkateswaran S., Govindan T.R.: *A Predictioned Navier-Stokes Method for Two-Phase Flows with Application to Cavitation Prediction*. American Institute of Aeronautics and Astronautics. 1999.

[8] Štefan D., Rudolf P.,Hudec M., Rudolf P.: *Numerical and experimental investigation of swirling flow in a conical diffuser.* EPJ web of conferences. 2015.

11. Acknowledgement

The research has been supported by project of the Czech Science Foundation No P101/13-23550S “Experimental research and mathematical modelling of unsteady phenomena induced by hydrodynamic cavitation”. Research was also supported by project FSI-S-14-2480 (Innovative fluid machines) of Brno University of Technology, Faculty of Mechanical Engineering