[1.]Jasmina BOGDANOVIĆ-JOVANOVIĆ, [2.]Živojin STAMENKOVIĆ, [3.]Živan SPASIĆ, [4.]Jelena PETROVIĆ, [5.]Miloš KOCIĆ

NUMERICAL INVESTIGATION OF CAVITATING FLOW IN VENTURI NOZZLE

[1-5.]Faculty of Mechanical Engineering, Niš, SERBIA

Abstract: Cavitation problems occurs in water flow where the local pressure reaches saturated vapour pressure, due to the sudden increase of local velocity values, forming bubbles and different forms of flow discontinuity. Water flow in the Venturi nozzle is the typical example of potentially cavitation flow. Due to the large importance of Venturi nozzles in engineering practice, it is important to determine their cavitation characteristics, such a cavitation number and loss coefficient of the nozzle. In case we are unable to obtain cavitation characteristics of the nozzle experimentally, there is the possibility to conduct numerical simulations of the flow in the Venturi nozzle. The objective of this study is to present the possibilities of obtaining cavitation characteristics of Venturi nozzles by applying numerical simulations, and to determine the optimal numerical modelling of flow in the case of Venturi nozzle.
Keywords: Venturi nozzle, numerical simulation, cavitation, cavitation number, Venturi loss coefficient

1. INTRODUCTION

Venturi effect is based on the sudden local flow velocity increase, due to the principle of mass continuity in the specific geometry of the nozzle, which leads to the local static pressure decrease, according to the low of mechanical energy conservation. The most common use of a Venturi tubes is in the measuring devices, such as Venturi meters used for measuring the volumetric flow rate of the liquids. The application of the Venturi tubes in the process industry and the industry in general is huge and significant, therefore the determination of their operating (and cavitation) characteristics are considerably important. Operating characteristic of Venturi nozzle depend on flow characteristics and geometry of the nozzle. The formation and development of cavitation in the Venturi strongly affect the operation of hydraulic system in which it is installed, usually as the limiting factor.

There are several phases in the development of cavitation in a convergent-divergent nozzle such as Venturi nozzles [7,8], from cavitation inception through partial cavitation, full cavitation and supercavitation. Generally with increasing local flow velocity a pressure decrease in the throat of Venturi nozzle until it reaches the value of saturation pressure when vapour bubbles firstly appear in the water (so called cavitation inception). The bubbles grow and move toward the region with higher pressure, where they collapse suddenly, usually in the area near solid surfaces, causing the undesired phenomena of cavitation erosion. With a further reduction of pressure, bubbles grow and increase their number, accumulating into pockets of vapour, affecting the entire flow domain downstream. Unexpectedly, the further development of cavitation reduces the surface damage of the Venturi nozzle. In the regime of supercavitation a larger scale bubbles are formed in the central part of the nozzle, forming a kind of bubble spray. The air bubbles collapses downstream in the central part of the Venturi nozzles, away from the wall surfaces. Many attempts have been made to fully clarify the mechanisms of cavitation generally [11] and in a Venturi nozzle [1]. However, considering such a nonstationary phenomenon with numerous influential factors, this topic is still of interest to the scientific community.

In previous research a lot of attention is devoted to experimental measurements in convergent-divergent nozzles [2,3]. Using sophisticated high speed cameras is enabled the visualization of cavitation and obtaining cavitation patterns for all cavitation regimes [2].

In order to overcome the lack of laboratory space and equipment, there were many attempts to conduct numerical simulation of fluid flow in Venturi nozzles, with emphasis on two-phase flow, which occurs due to cavitation [4,5,9,12].

2. CAVITATION PARAMETERS

For characterization of the cavitation usually is used so called cavitation number, represented by the formula:

$$\sigma = \frac{p_2 - p_v}{\rho\left(v_{th}^2 / 2\right)} \tag{1}$$

where p_2 is downstream pressure, p_v is the pressure of the vapour phase and v_{th} is the velocity in the throat of

the Venturi nozzle.
Occasionally, a cavitation index K_c is used, mostly for studying cavitation in valves [13], or sometimes its reciprocated value ε:

$$\varepsilon = 1/K_c = (p_1 - p_v)/(p_1 - p_2) \tag{2}$$

There are many empirical formulas for obtaining a saturated vapour pressure, depending on the temperature. For temperatures above 0^0C, some of well-known formulas are:

- Tetens' formula: $$p = 0{,}61078 \cdot e^{\frac{17{,}27 \cdot t}{t+237{,}3}} \tag{3}$$
- Arden Buck formula: $$p = 0{,}61121 \cdot e^{\left(18{,}678 - \frac{t}{234{,}5}\right)\left(\frac{t}{257{,}14+t}\right)}, \tag{4}$$

where p – pressure [kPa], t – temperature [°C].
For water temperature t=25°C, a value of saturated vapour pressure is $p_v \approx 3168$Pa.
Energy dissipation in Venturi nozzle can be observed using Venturi loss coefficient defined by the formula:

$$\zeta = \frac{p_1 - p_2}{\rho\left(v_{th}^2/2\right)} \tag{5}$$

Dimensionless parameters allow more universal classification and and better overview of cavitation regimes.

3. GEOMETRY AND DISCRETIZATION MESH OF VENTURI NOZZLE

Venturi nozzle is an example of convergent-divergent nozzle. A geometry of studied Venturi nozzle is shown in Figure 1.
Discretization mesh is made using software ICEM CFD 14.0. For the purpose of this investigation, there are few meshes created, in order to obtain the optimal one. For obtaining the optimal mesh, the grid independence test have been conducted.

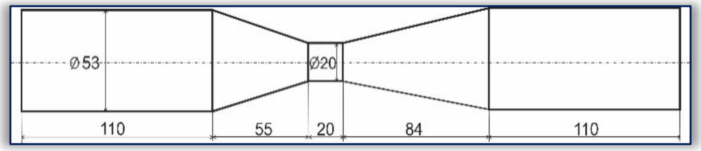


Figure 1. Geometry of Venturi nozzle

4. GRID INDEPENDENCE TEST

To reduce the computational time and to ensure satisfactory accuracy of the results at the same time, different discretization meshes are tested, shown in Table I. The mesh number 2 in the Table I has been chosen, obtaining result error around 3% for the operating regime. The mesh is consists of 1056506 elements (565281 tetrahdra and 491225 wedges), shown in Figure 2.

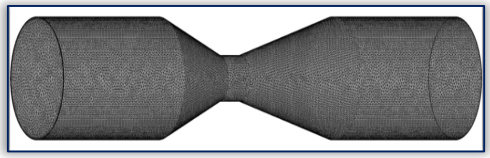


Figure 2. Discretization mesh of Venturi nozzle

Table 1 Grid independence test results ($\dot{m}$=9 kg/s, k-turbulence model)

Number of mesh elements	y^+ average	Cavitation number σ	Relative Error [%]
~ 600000	47.7	0.253	5.3
~ 1280000	30.9	0.247	3.0
~ 2270000	17.4	0.245	2.2

For all three created meshes, the maximal value of y^+ is around 100. For the chosen mesh with approximately 1280000 elements, in the central longitudinal plane the average value of y^+=17.4, which is considered as a good quality mesh [10].

5. NUMERICAL SIMULTION OF FLUID FLOW IN VENTURI NOZZLE

The two phase numerical simulations of water flow in the Venturi nozzle are conducted using Ansys CFX 14.0 for different flow regimes. The results are compared with eperimental results [2], quantitatively and qualitatively. RANS numerical simulations are usually the first choice, considering the time and computational resources. In such highly non-stationary problems, it has been shown qualitatively good results obtained by RANS numerical simulations, eventhough the qualitatively the fluid flow pattern changes in time (the flow is unsteady). Four different turbulence models have been used (k-ε, k-ω, BSL Reynolds Stress and Shear Stress Transport - SST). The best results are obtained using SST model. Further on is determined functional dependence σ-ζ, shown in Figure 3. Rayleigh Plesset cavitation model is used for modeling dynamics of water bubbels in the cavitation regimes. As an advection scheme has been used the high-resolution scheme.
Further, numerical investigations of ustationary nature of fluid flow in the Venturi nozzle have been conducted. Some qualitatively better results are obtained using URANS approach (i.e., transient numerical simulations). Detached Eddy Simulations, as the hybrid model, and, finally LES numerical simulations (Smagorinski) were used as well. Results obtained for the same flow regime (mass flow rate is 9 kg/s) and compared to experimental results [2]. All unsteady siulations obtained higly accurate values for cavitation number, Venturi loss coefficient and

cavitation index. URANS and DES simulation results are very similar (maximal relative error 2% for the operating regime $\dot{m}$ =9 kg/s), while LES simulation provides even more accurate results, with a relative error less then 1% for all coefficients. Unsteady numerical simulations offer a much better insight into the cavitation pattern. Fluid flow patterns evidently vary with numerical approaches. In Figure. 4 and Figure.5 is presented volume fraction of water vapor for different flow regimes.

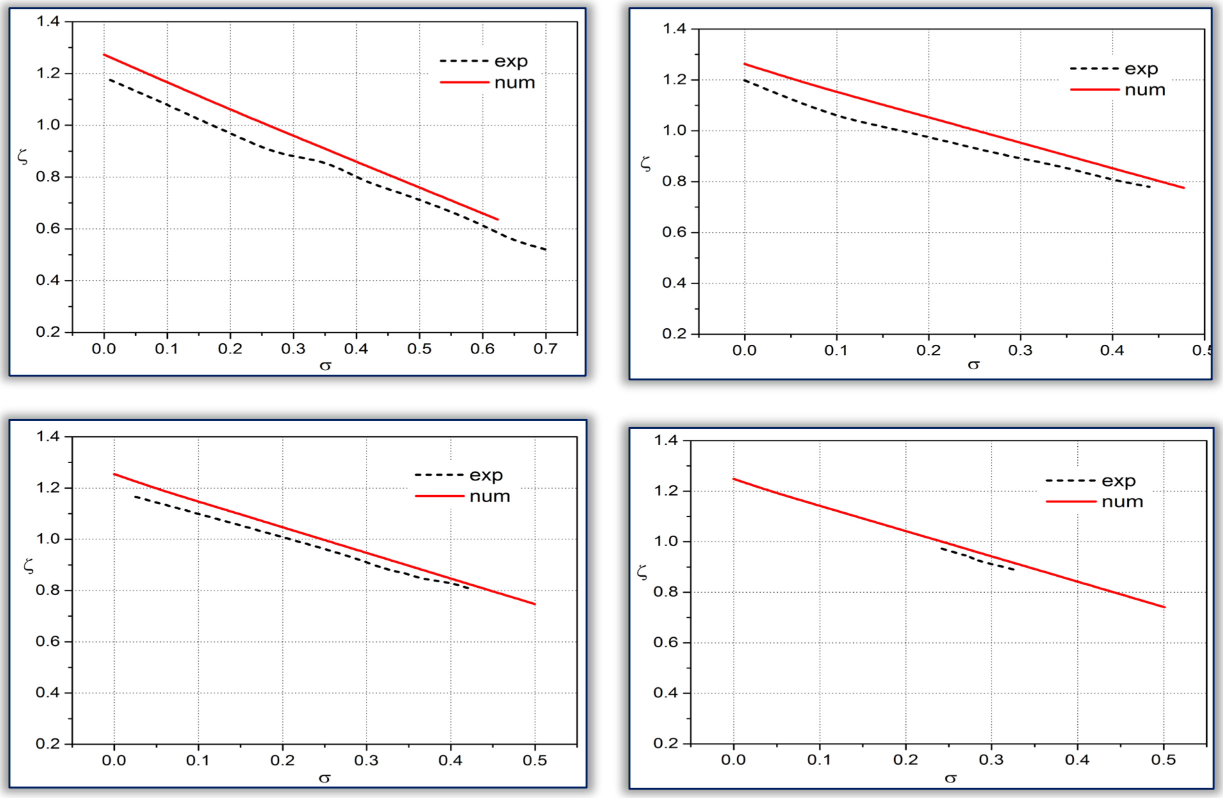
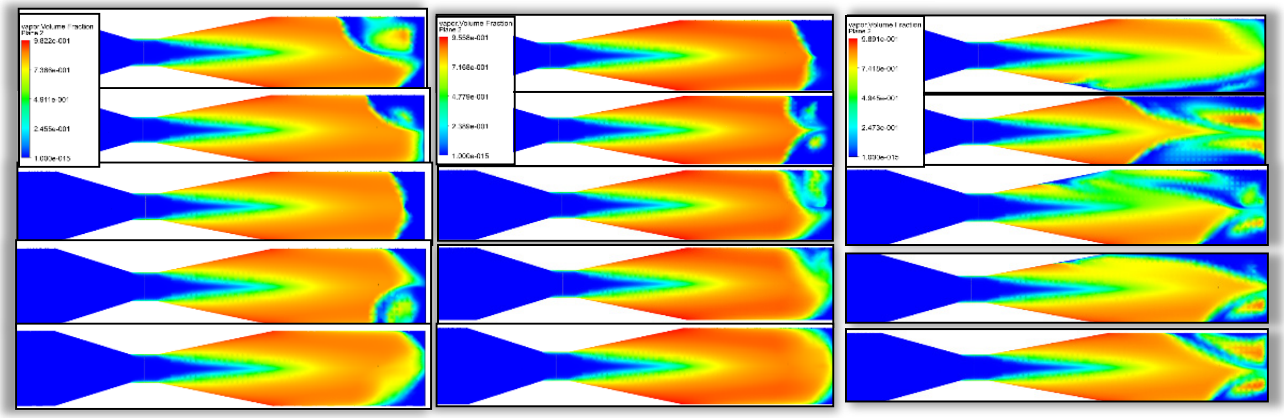


Figure 3. σ-ζ diagrams for different mass flow rates: a) $\dot{m}$ = 7 kg/s , b) $\dot{m}$ = 8 kg/s ,
c) $\dot{m}$ = 9 kg/s and d) $\dot{m}$ = 10 kg/s .



a) b) c)

Figure 4. Vapor volume fraction in longitudinal section: a) URANS, b) DES, c) LES

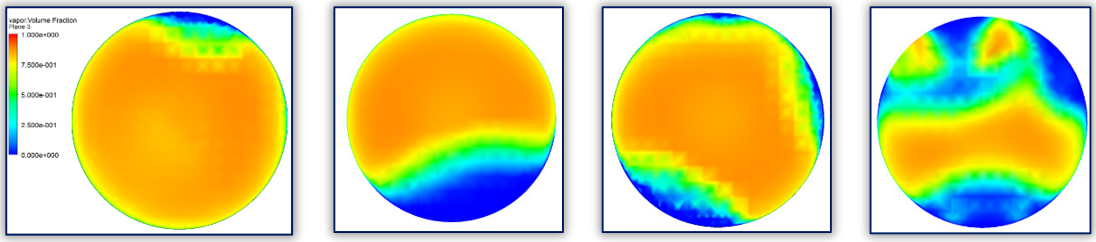


Figure 5. Vapor volume fraction in cross-section of the Venturi nozzle (z=0.35 m):
a) RANS (SST), b) URANS, c) DES, d) LES

6. CONCLUSIONS

Compared with available experimental results, numerical simulations of the water flow in the Venturi nozzle show quantitatively good results for various operating regimes (i.e. various water flow rates). Very accurate results of obtained pressure and velocity are provided even by RANS numerical simulations The average relative deviations of the obtained cavitation numbers depends on the Venturi flow rate, and it is only 2.8% for the flow rate of 10 kg/s, while the maximum average relative deviation of cavitation numbers is 8% for the flow rate of 7 kg/s. However RANS simulation cannot provide satisfactory results when it comes to the resulting flow field in cavitation regimes, particularly turbulent fluctuations in the output part of the Venturi nozzle, nor to predict cavitation patterns.

In case one wants to examine the flow pattern in Venturi nozzle and to obtain a proper visualization of the cavitation regime transition, an unsteady numerical simulation of fluid flow is essential. In this paper URANS, DES and LES approach for one flow regime are compared, giving priority to the first method due to the shorter time required for the simulation. But to achieve greater accuracy, especially when it comes to obtaining fluctuating velocity field and cavitation patterns, priority must always be given to LES simulations.

Note

This paper is based on the paper presented at 13th International Conference on Accomplishments in Mechanical and Industrial Engineering – DEMI 2017, organized by University of Banja Luka, Faculty of Mechanical Engineering, in Banja Luka, BOSNIA & HERZEGOVINA, 26 - 27 May 2017.

References

[1] Charrière B., Decaix J., Goncalvès E., (2015). A comparative study of cavitation models in a Venturi flow, European Journal of Mechanics B/Fluids 49, 287–297.
[2] Rudolf P., Hudec M., Gríger M., Štefan D. (2014), Characterization of the cavitating flow in converging-diverging nozzle based on experimental investigation, EPJ Web of conferences, 67.
[3] Niedzwiedzka A., Sobieski V., (2016). Experimental investigation of caviting flows in a Venturi tube, Technical Sciences, 19(2) 151–164.
[4] Hari Vijay P., Subrahmanyama V., (2014). Cfd simulation on different geometries of venturimeter, IJRET: International Journal of Research in Engineering and Technology, Volume: 03, Issue:07, 456-463.
[5] Tamhankar N., Pandhare A., Joglekar A., Joglekar A., (2014). Experimental and CFD analysis of flow through venturimeter to determine the coefficient of discharge, International Journal of Latest Trends in Engineering and Technology (IJLTET), Vol. 3 Issue 4, 194-200.
[6] Koukouvinis P., Naseri H., Gavaises M., (2016). Performance of turbulence and cavitation models in prediction of incipient and developed cavitation, International J of Engine Research, 1-18.
[7] Dixon S. L., (1998). Fluid Mechanics and Thermodynamics of Turbomachinery, Elsevier Inc., 4th edition.
[8] Knapp R. T., Daily J. W., Hammitt F. G., (1970). Cavitation, McGraw-Hill.
[9] Frenander K., (2014). Optimization and Automated Parameter Study for Cavitating Multiphase Flows in Venturi - CFD Analysis, Master's Thesis, Department of Applied Mechanics, Chalmers University of Technology.
[10] Ferziger, J. H., Peric, M.,(2002). Computational methods for fluid dynamics, third rev edition, Springer.
[11] Brennen C. E., (2011). An Introduction to Cavitation Fundamentals, invited lecture, Cavitation: Turbo-machinery & Medical Applications, WIMRC FORUM 2011, University of Warwick, UK.
[12] Kozak J., Rudolf P., Stefan D., Hudec M., Griger M., (2015). Analysis of pressure pulsation of cavitating flow in converging-diverging nozzle, 6th IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Ljubljana, Slovenia.
[13] Samson A.G.,Cavitation in control valves, from http://www.samson.de

ISSN 1584 - 2665 (printed version); ISSN 2601 - 2332 (online); ISSN-L 1584 - 2665