Journal of the Technical University of Gabrovo
https://mc04.manuscriptcentral.com/jtug

CFD MODELING OF HYDRODYNAMIC CAVITATION IN A VARIABLE THROTTLE

Ihor Nochnichenko[1*], Dmytro Kostiuk[2], Oleksandr Haletskyi[2], Stas Serhiy[3]
[1] Jagiellonian University, W.M., Krakow, Poland
[2] National Technical University of Ukraine "Igor Sikorsky Kyiv Polytechnic Institute", Kyiv, Ukraine
[3] National University of Civil Protection of Ukraine

ARTICLE INFO

Article history:
Received 6 October 2025
Revised 14 November 2025
Accepted 19 November 2025

Keywords:
CFD, hydrodynamic cavitation, variable throttle, multiphase flow, turbulence modeling

http://doi.org/10.62853/FCQW4829

ABSTRACT

This study presents a computational fluid dynamics (CFD) investigation of hydrodynamic cavitation in a variable throttle. The primary objective is to analyze the onset, growth, and suppression of cavitation phenomena under different throttle openings and flow conditions, including variations in inlet pressure, velocity, and temperature.

The simulations were conducted using the turbulence model combined with a cavitation model accounting for liquid - vapor phase change. Several throttle configurations were examined to evaluate local pressure distribution, vapor volume fraction, cavitation zone development, and associated hydraulic losses.

The results indicate that at small throttle openings cavitation remains weak and localized near the vena contracta, while increasing the degree of throttling significantly expands the cavitation region and can lead to local pressure drops below the vaporization threshold. This causes the formation of extended vapor pockets with potential for material erosion. Additionally, fluid temperature and liquid properties (viscosity, vapor pressure) strongly influence the critical conditions for cavitation onset.

Overall, the findings highlight that proper throttle design - aimed at reducing regions of critical pressure - and optimized control of throttling degree and flow parameters can mitigate cavitation intensity and its detrimental effects. These results provide useful insights for the design and operation of hydraulic systems, pumps, injectors, and turbines where variable throttling is commonly employed.

1. INTRODUCTION

Hydrodynamic Cavitation (HC) is a complex physical phenomenon that occurs when the local pressure of a fluid drops below its saturated vapor pressure, leading to the formation, growth, and subsequent rapid collapse of vapor bubbles [1]. This phenomenon can be both detrimental and beneficial.

In hydraulic systems, uncontrolled cavitation is a primary cause of reduced efficiency in pumps, valves, and hydroturbines, inducing vibration, noise, and, most significantly, cavitation erosion of surfaces, which can ultimately lead to equipment failure [2, 3].

On the other hand, controlled HC is increasingly applied as a highly efficient method for process intensification. Furthermore, during the adiabatic collapse of cavitation bubbles in a liquid, extremely high local temperatures (up to 1000 K and above) and pressures are generated, leading to the formation of free radicals and, under certain conditions, sonoluminescence (light emission) [4, 5]. This unique property is utilized for water treatment, disinfection, homogenization, and chemical reactions [6].

The regulation and optimization of this process require the use of throttling devices with variable flow cross-sections (e.g., needle valves, adjustable diaphragms). These elements allow for the dynamic control of the pressure drop and, consequently, the intensity of cavitation. However, the complex interplay between the dynamic change in geometry and the unsteady, multiphase flow makes predicting system behavior extremely difficult.

Traditional analytical and empirical models are insufficient for accurately describing the local characteristics of cavitating flows due to their three-dimensional and unsteady nature. Consequently, Computational Fluid Dynamics (CFD) has become an indispensable tool for studying HC.

Historically, most CFD studies have focused on modeling cavitation in elements with static geometry, such as injector nozzles [7], Venturi tubes [8], or simple orifices [9]. It has been demonstrated that the application of the $k-\omega$ SST turbulence model combined with multiphase models (e.g., the Mixture Model) and mass transfer models, such as the Zwart-Gerber-Belamri (ZGB) model, yields the best results in predicting the length and shape of the cavitation cloud [10].

---

[*] Corresponding author. E-mail: ihor.nochnichenko@uj.edu.pl

The problem under consideration lies in the lack of detailed numerical studies that correlate the dynamics of the throttling device's variable flow cross-section with the velocity of the cavitation front propagation and the total vapor phase volume. A precise understanding of this relationship is critical for the development of highly efficient regulating valves and new-generation cavitation reactors that require rapid and accurate control over HC intensity.

2. EXPOSITION

Numerical modeling conducted using the integrated CFD package SolidWorks Flow Simulation 2025. This tool employs the Finite Volume Method (FVM) and effectively handles the automatic generation of a mesh that adapts to the complex geometry of the throttling device.

Most engineering fluid flows are turbulent, so Flow Simulation is primarily designed to model such conditions. It uses Favre-averaged Navier-Stokes equations, considering time-averaged turbulence effects while directly accounting for large-scale, time-dependent phenomena. This introduces Reynolds stresses, which require extra information. To close the system, Flow Simulation applies the k-ε model, using transport equations for turbulent kinetic energy and its dissipation rate. The software uses one system of equations for both laminar and turbulent flows, allowing transitions between these states [11].

Cavitation occurs when a liquid's pressure drops below its saturation pressure at a given temperature, causing vapor-filled cavities to form. If vapor forms much faster than the liquid flows, the process approaches thermodynamic equilibrium, allowing for equilibrium-based simulation of the phase transition. In this Flow Simulation was used the equilibrium cavitation model. This homogeneous equilibrium model applies to water and analyzes two-phase flows, including thermal effects like localized boiling [11].

To model hydrodynamic cavitation within the cavitator, a solid-body model was constructed with the dimensions shown in the Fig. 1 [12].

The throttling gap is formed by a 0.8 mm diameter bore in the PMMA (Polymethyl methacrylate) body and the end face of the screw head (simplified representation shown on the Fig. 3). In the experiment, the throttling gap area was approximately 0.25 mm^2, with the screw head occluding half of the bore.

Water at a temperature of 20 C (293 K) was used as the model substance.

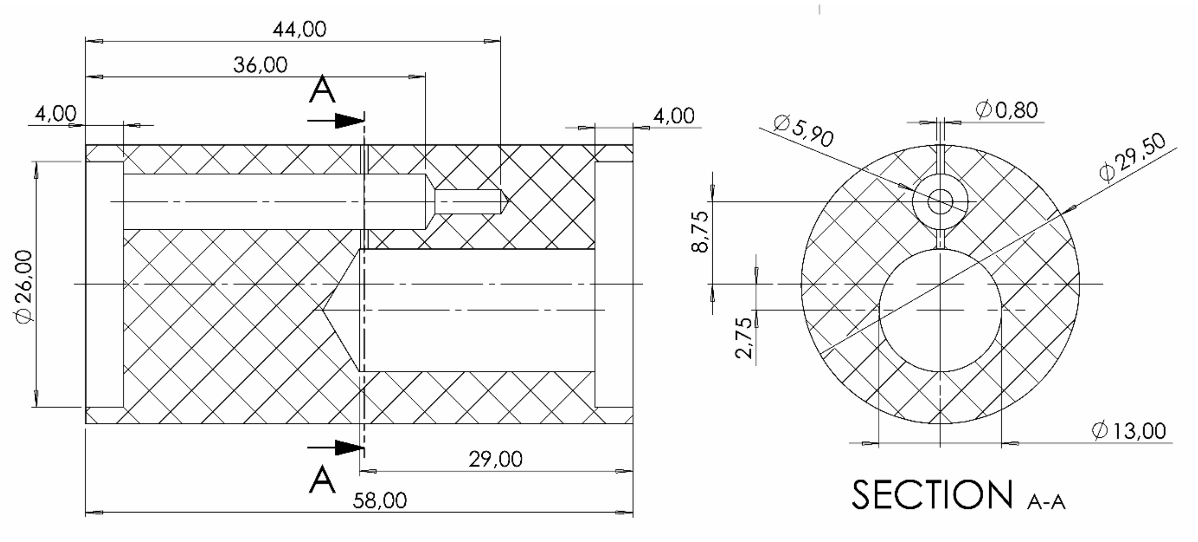

Fig. 1. Dimensions of the PMMA cavitator body

The generated computational mesh was created taking into account the channel geometry, featuring fine elements in the region of the throttling gap and coarser elements in the cavitator's inlet and outlet channels. The total number of fluid mesh elements is 70,998, with 14,206 fluid elements contacting the surface. Computational mesh for simulation is shown on Fig. 2.

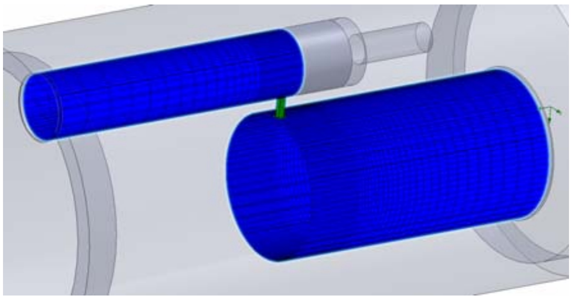

Fig. 2. Computational domain and meshing

As boundary conditions, a gauge pressure of 5,5 MPa was set at the cavitator inlet, and atmospheric pressure was set at the outlet. The boundary conditions are shown on the Fig. 3.

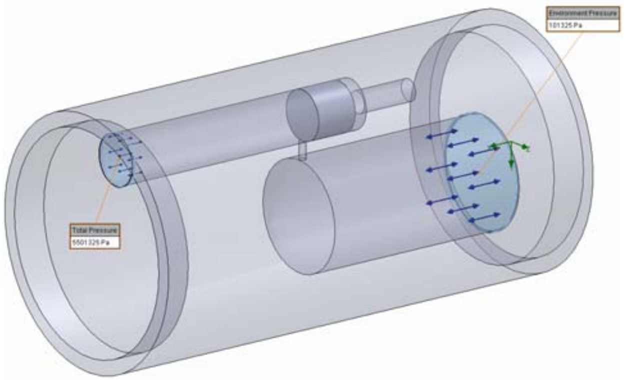

Fig. 3. Three-dimensional model of the cavitator and boundary conditions

As a result of the simulation, contours of velocity, pressure, temperature, fluid density, and vapor mass fraction within the cavitator channels were obtained. The results are shown in the Fig. 4 - 8.

This makes it possible to attempt a comparison between the visualized cavitation observed in the physical experiment [12] and the simulation results.

As demonstrated by the obtained velocity distributions, the maximum fluid velocity under these conditions reaches 102 m/s. The region of maximum velocity coincides with the zone where hydroluminescence was observed in the physical experiment.

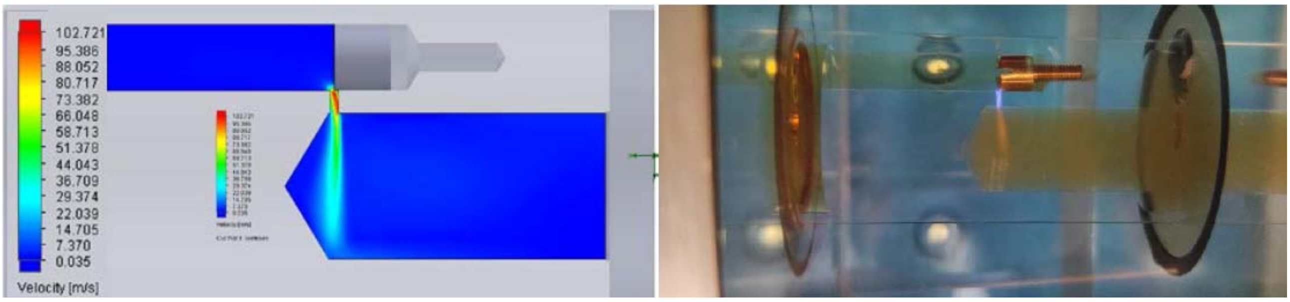

Fig. 4. Velocity distribution (left) and flow visualization obtained by physical experiment (right)

Analyzing the pressure distribution map (Fig. 5), a region of significant pressure drop (blue region, up to 2 kPa absolute pressure) and cavity formation can be observed.

This is further corroborated by the contours showing changes in fluid density and vapor mass fraction.

An analysis of the change in the working fluid's temperature indicates that the temperature increases by approximately 3 K after passing the throttling gap (see Fig. 6) [14, 15]. This thermal rise closely aligns with the theoretical calculations based on the methodology presented in the source.

The temperature rise for the fluid across the orifice:

$$\Delta T = \frac{\Delta p_{OR}}{\rho . c_p},$$

where Δp_{OR} - the pressure drop across the orifice = 55 bar; the fluid density $\rho = 1000\, kg/m^3$ and the specific heat coefficient $c_p = 1.8\, kJ/kg.K$.

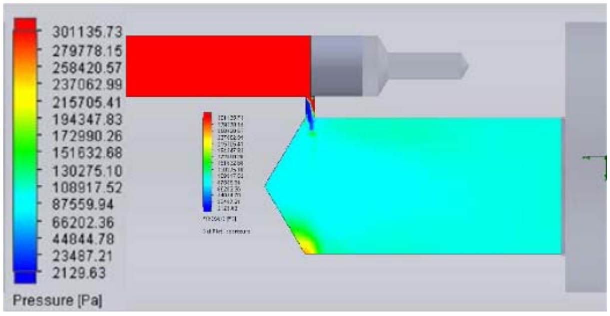

Fig. 5. Pressure distribution in the cavitator

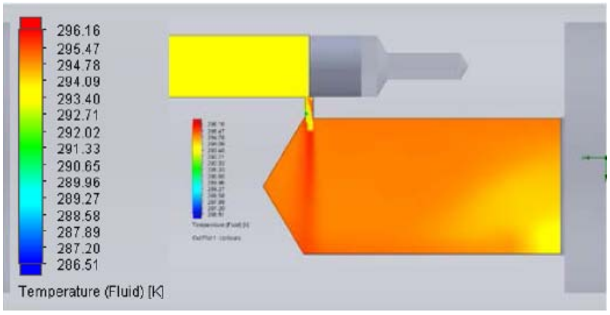

Fig. 6. Fluid temperature distribution in the cavitator

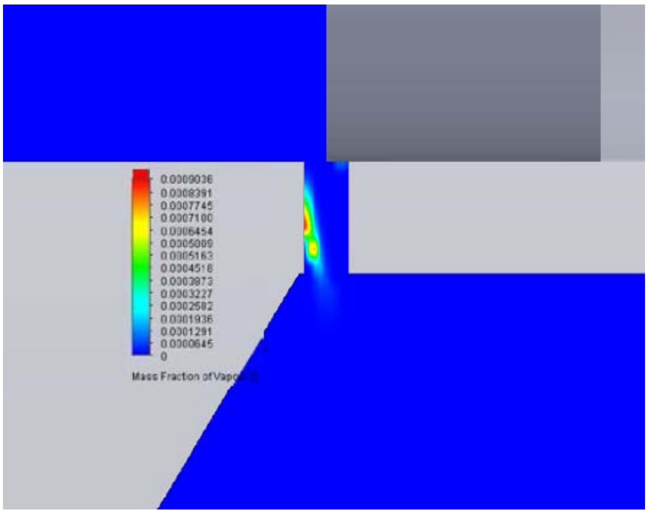

Fig. 7. Mass fraction of vapor distribution in the throttling gap

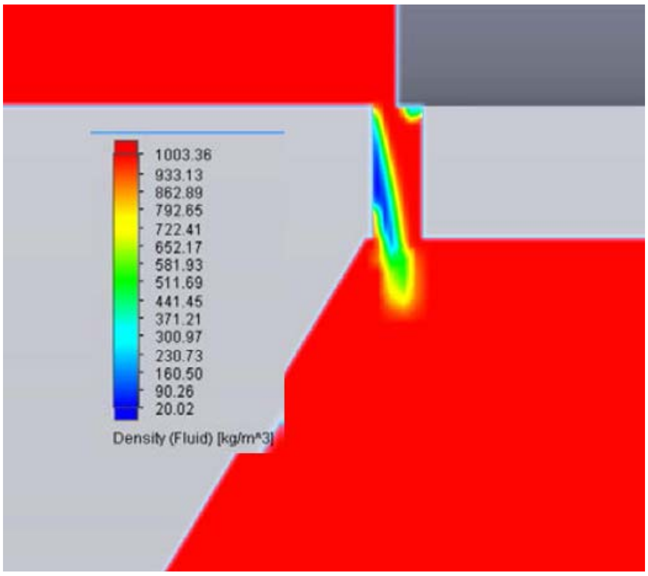

Fig. 8. Fluid density distribution in the throttling gap

3. CONCLUSION

In this study, CFD simulation was performed for a variable throttle to validate whether the numerical modeling available in Solidworks Flow Simulation predicted reliably and accurately the complex flow inside the orifice. Considering the complex nature of the hydrodynamic cavitation phenomenon, particularly hydroluminescence, there is a need for a tool to help determine the conditions under which hydroluminescence occurs and the effect of operating parameters on its intensity.

As a first approximation, it can be concluded that this software package allows for obtaining a flow pattern similar to that observed during the physical experiment.

Further research is planned to focus on determining the effect of the working fluid type on cavity formation and identifying the conditions under which hydrodynamic cavitation is observed in this type of throttling device.

REFERENCES

[1] Brennen, C.E., Cavitation and Bubble Dynamics, Oxford University Press (2014)

[2] Knapp R.T., Daily J. W., Hammitt F. G., Cavitation, McGraw-Hill (1970)

[3] Wang J., Sun L., Zhou Y., Liu Y., Zhao F., Numerical Simulation of Cavitation Characteristics of a Centrifugal Pump Based on an Improved ZGB Model, Processes 11(2) (2023) 438, https://doi.org/10.3390/pr11020438

[4] Suslick K.S., Sonochemistry. Science 247(4949) (1990) 1439–1445

[5] Arrojo S, Benito Y., A theoretical study of hydrodynamic cavitation, Ultrasonics sonochemistry 15 (2008) 203-11, 10.1016/j.ultsonch.2007.03.007

[6] Carpenter J., Badve M., Rajoriya S., George S., Saharan V., Pandit A., Hydrodynamic cavitation: An emerging technology for the intensification of various chemical and physical processes in a chemical process industry, Reviews in Chemical Engineering (2016) 33. 10.1515/revce-2016-0032

[7] Edelbauer W., Numerical Simulation of Cavitating Injector Flow and Liquid Spray Break-up by Combination of Eulerian-Eulerian and Volume-of-Fluid Methods, Computers & Fluids (2016) 144. 10.1016/j.compfluid.2016.11.019

[8] Lee G., Jhung M., Bae J., Kang S., Numerical Study on the Cavitation Flow and Its Effect on the Structural Integrity of Multi-Stage Orifice, Energies 14(6) (2021) 1518, https://doi.org/10.3390/en14061518

[9] Olivier Coutier-Delghosa R., Fortes P., Jean-Luc R., Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation, Journal of Fluids Engineering 125 (1) (2003) 38-45, ff10.1115/1.1524584ff. ffhal-00211202f

[10] Zwart P., Gerber A.G., Belamri T., A two-phase flow model for predicting cavitation dynamics, Fifth International Conference on Multiphase Flow (2004)

[11] Technical reference SOLIDWORKS FLOW SIMULATION (2025)

[12] Nochnichenko I., Luhovskyi O., Kostiuk D., Jakhno O., Research of the Influence of Hydraulic Orifice Material on the Hydrodynamic Cavitation Processes Accompanied by Luminescence (2021) 10.1007/978-3-030-59509-8_26

[13] Vacca A., Franzoni G., Hydraulic Fluid Power: Fundamentals, Applications, and Circuit Design, 1st edition, John Wiley & Sons, Hoboken, New Jersey (2021)

[14] Nochnichenko I., Kostiuk D. Kryvosheiev V., Experimental Study of the Effect of a Magnetic Field on Hydrodynamic Cavitation Accompanied by Luminescence (Advances in Hydraulic and Pneumatic Drives and Control, Centrifugal Pumps, Valves, and Seals (NSHP 2025) 160-168, https://link.springer.com/chapter/10.1007/978-3-032-07392-1_16

[15] Nochnichenko I., Luhovskyi A., Jakhno O., Kostiuk D., Komada P., Kozbakova A., Experimental research of hydroluminescence in the cavitating flow of mineral oil, Photonics Applications in Astronomy, Communications, Industry, and High-Energy Physics Experiments 2019, Wilga, Poland 11176 (2019) 1117615, https://doi.org/10.1117/12.2536946