

Numerical Design and Experimental Verification of a Novel Cavitation Device Based on Venturi Effect in Fluid Mechanics

Shangxuan Chen^{1,a,*}

¹Shanghai Pinghe Bilingual School, Huangyang Road, Shanghai, China

a. chenshangxuan@shphschool.com

**corresponding author*

Abstract: As a complex hydrodynamic phenomenon, cavitation has always been a focus of attention due to its harmful effects. However, it is important to note that the collapse of cavitation bubbles produces instantaneous high temperatures and pressures as well as high-speed jet phenomena, which means this process contains tremendous energy. This energy can not only enhance physical and chemical processes but also achieve the goals of energy conservation and efficiency improvement. Currently, there have been no breakthroughs in effectively generating cavitation, adopting highly efficient cavitation devices, or accurately quantifying and analyzing the effects of cavitation. Therefore, research on the evolution process of cavitation and the factors influencing its effects is significant. In recent years, cavitation technology has developed rapidly in the field of wastewater degradation. To improve the efficiency of cavitation degradation, based on the theory of the Venturi effect in fluid mechanics, this project proposes a novel cavitation generator using a narrow slit type Venturi tube through a combination of motion equations and computational simulation. This design aims to increase shear area and enhance the yield of cavitation bubbles. A low-cost, low-energy consumption new cavitation device was optimized and manufactured based on numerical simulation. The new cavitation generator was used to explore the influence of different throat geometric characteristics and expansion angles on cavitation capability. Firstly, the continuity equation and momentum equation were derived based on Rayleigh's cavitation theory and the Venturi effect in fluid mechanics; secondly, a computational fluid dynamics (CFD) grid was established and finite element simulation calculations were conducted for the structural optimization design of the new cavitation device; finally, by combining fluid mechanics theoretical formulas and numerical simulation, a low-cost, low-energy consumption new cavitation device was manufactured, and experimental analysis was conducted to study the influence of different throat geometric characteristics and expansion angles on cavitation capability.

Keywords: Venturi cavitation, Venturi effect, Computational fluid dynamics, Experimental verification

1. Introduction

With the rapid development of modern industry, the extensive use of strong fertilizers, pesticides, dyes, and their organic intermediates has led to an increase in the types and quantities of refractory organic pollutants in industrial wastewater and natural water bodies. These organic pollutants, due to their biotoxicity and recalcitrance, can persist in the environment for a long time, causing various acute and chronic impacts on organisms, becoming one of the global focuses of concern.[1]

Traditional water treatment methods have become increasingly ineffective in degrading these complex wastewater components. As an advanced oxidation process, cavitation technology has been proven to effectively degrade aromatic hydrocarbons, heterocyclic compounds, and phenols in wastewater. Cavitation occurs when the local pressure at a certain point in the liquid is lower than the saturated vapor-liquid separation pressure, resulting in the formation of many gas bubbles in the liquid. These bubbles move with the liquid flow to a high-pressure region where they collapse under the pressure gradient. The entire process of the bubbles moving from the low-pressure to the high-pressure region and collapsing is known as cavitation. Studies have shown that cavitation can produce temperatures above 5000K and pressures around 50MPa locally, accompanied by strong shock waves and jets traveling at speeds up to 400km/h. This energy from cavitation can break the hydrogen bonds between water molecules, decomposing water into hydroxyl and hydrogen radicals, and producing other free radicals and a certain amount of charged particles in the liquid.

The object of this study is shown in Figure 1., where the Venturi tube consists of a contraction section, a throat, and an expansion section. Fluid will generate a local low-pressure zone at the throat, and when the local pressure is lower than the saturated vapor pressure, gas bubbles and steam will be produced. As the fluid further flows downstream, due to the increased pressure in the expansion section, these bubbles will eventually collapse in the expansion section. This is a brief overview of the cavitation process in a Venturi tube.

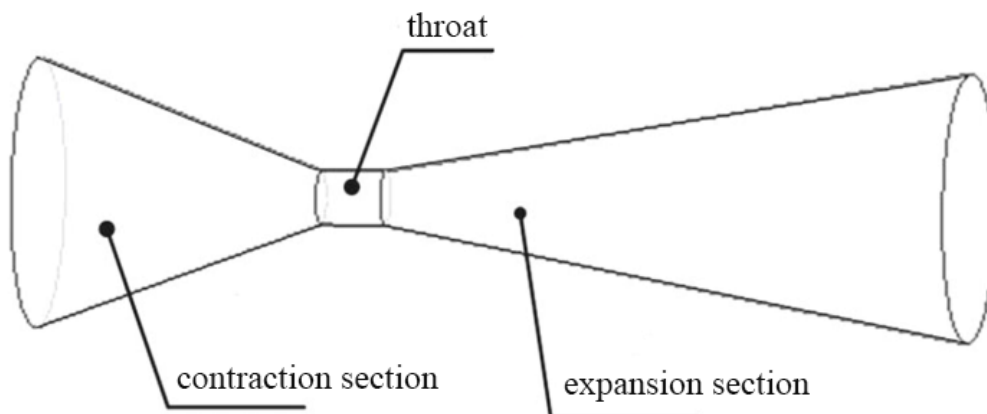


Figure 1: Structure of Venturi Tube

There are three modes of bubble collapse in a Venturi tube: The first mode is collapse in the pressure gradient region. Due to the presence of a pressure gradient within the region, the shape of the bubble becomes flat under pressure, and finally, it collapses under the action of the cavitation water jet. The second mode is the collapse of a bubble on a solid wall. During this process, the side of the bubble that does not contact the solid wall is affected by the pressure, and the middle part gradually sinks and penetrates until it collapses. The third mode is a situation between the first and second collapse modes. The bubble does not adhere to the wall but is close to it and then collapses.

Researchers from Zhejiang University of Technology, such as Dong Zhiyong, Zhang Shaohui, and Yang Jie, developed a combined porous plate and Venturi hydrodynamic cavitation reaction device and studied the influence of throat flow velocity and hole arrangement on sterilization effects [2]. Liu Yanan from Harbin University of Science and Technology studied the cavitation effects of serial and single-stage Venturi tubes and concluded that the cavitation effect of single-stage Venturi tubes is superior to that of serial Venturi tubes [3]. Rayleigh established the motion equation for spherical cavitation bubbles and solved the collapse of an ideal liquid cavity in a homogeneous region based on the principle of energy balance, laying the foundation for the dynamics theory of cavitation bubbles [4]. Wang Zhiyong from Dalian University of Technology used FLUENT software to perform numerical simulations on a Venturi tube, and the results showed that an increase in the fluid temperature inside the pipe will increase the saturated vapor pressure of the liquid, which is beneficial for cavitation. However, an increase in temperature will also reduce the internal gas content, thereby inhibiting the cavitation effect [5].

The main research content of this topic is to study the evolution process and influencing factors of hydraulic cavitation in a Venturi tube. The specific research content includes: deriving the dynamic two-phase cavitation model and the pressure loss model inside the Venturi tube, using CFD software FLUENT for numerical simulation, analyzing the influence of different factors on the cavitation effect, and finally conducting experimental exploration and research.

The core research content includes:

Cavitation Mechanism Inside the Venturi Tube: Analyze the dynamic changes of gas-liquid two-phase flow inside the Venturi tube, including the mechanisms of bubble formation, collapse conditions, and regions where cavitation occurs. Starting from macrofluids, obtain the control equations for fluid flow inside the Venturi tube by deriving the mass, momentum, and energy conservation equations. Then, starting from a microscopic perspective, establish a bubble dynamics model and consider the shortcomings of steady-state cavitation models, establishing a dynamic cavitation model for gas-liquid two-phase flow inside the Venturi tube, i.e., performing theoretical cavitation modeling to determine the key factors affecting the cavitation effect in the Venturi tube.

Influencing Factors of Cavitation in the Venturi Tube: First, use DesignModeler to construct the geometric model of the Venturi tube fluid domain, and then use FLUENT software to mesh the established Venturi tube fluid domain model. By selecting appropriate cavitation models, fluid turbulence models, and multiphase flow models, and changing inlet and outlet parameters (inlet pressure, velocity, temperature, etc.) and different structural parameters (inlet cone angle, outlet cone angle, throat diameter, etc.), simulate and analyze the cavitation flow field inside the Venturi tube to draw conclusions.

Overall, through the above research, it aims to deeply understand the mechanism and influencing factors of cavitation phenomena in the Venturi tube, providing scientific basis and engineering practice guidance for optimizing the design of Venturi tubes.

2. Cavitation Theory and Control Equations

2.1. Cavitation Theory

Cavitation occurs because when the liquid flows through the throat of the Venturi tube, a local low-pressure zone is generated. The fluid transitions from laminar flow to turbulent flow. When the local pressure drops below the local saturated vapor pressure, combined with the unevenness of the internal pressure of the liquid, tiny gas nuclei appear in the liquid, gradually expanding and growing until the pressure inside the bubble exceeds the pressure of the liquid, at which point the bubble collapses. For a Venturi tube, when the fluid passes through the contraction section of the Venturi tube reactor, the pressure decreases with the increase in fluid velocity. Typically, once the pressure drops to the

saturated vapor pressure of the liquid, vaporization begins, producing cavitation bubbles similar to steam (sometimes, due to turbulence-induced pressure fluctuations, this can occur even when the pressure is higher than the vapor pressure). The cavitation bubbles grow as the pressure continues to decrease until they reach the throat of the Venturi tube. Then, with the presence of the diffuser section, the pressure gradually recovers, leading to the collapse of the cavitation bubbles and the creation of local hot spots, with some of the energy being released in the form of permanent pressure drop. The magnitude of the pressure drop greatly affects downstream cavitation and turbulence intensity.

According to the geometric shape of the Venturi tube, it can be divided into round Venturi tubes, slit Venturi tubes, and elliptical Venturi tubes. Research on annular and slit Venturi tubes is most common. The geometric parameters of the Venturi tube significantly affect cavitation behavior. Therefore, it is necessary to optimize the operating and geometric parameters of the Venturi tube to achieve the desired cavitation intensity. Many research reports have optimized geometric parameters of the Venturi tube based on divergence angles, the ratio of throat cross-sectional diameter to throat length (for slit Venturi tubes, the ratio of slit height to length), and the ratio of throat perimeter to opening area.

1. Bubble Generation and Growth

Cavitation number is an adimensional quantity used to describe the characteristics of cavitation effects. Its expression is:

$$C = \frac{p - p_v}{\frac{1}{2} \rho v^2}$$

From the expression of the cavitation number, it mainly affects liquid flow velocity, pressure, and density. Using the principle of energy conservation, Rayleigh transformed complex cavitation into solving the collapse of a spherical ideal cavity in an infinite homogeneous, incompressible, and non-viscous liquid, obtaining the famous cavity motion control equation:

$$\ddot{R}R \left(1 - \frac{R}{c}\right) + \frac{3}{2} \left(1 - \frac{\dot{R}}{3c}\right) R^2 = \left(1 - \frac{R}{c}\right) + \frac{1}{\rho} [p_B - p_{vt}] + \frac{R}{\rho c} \frac{d}{dt} [p_B - p_{vt}]$$

2. Bubble Collapse

When the local pressure is lower than the local saturated vapor pressure, combined with the unevenness of internal liquid pressure, tiny gas nuclei appear in the liquid and slowly expand. When the pressure inside the bubble exceeds that of the liquid, the bubble collapses. Liquid cavitation is a gradual growth process. The critical pressure for bubble collapse during collapse is:

$$p_0 = p_g \left[\frac{p_\infty (\gamma - 1)^{\frac{\gamma}{\gamma-1}}}{p_g} \right]$$

Professor Knapp from Caltech University explained three modes of bubble collapse: The first mode is collapse in a pressure gradient region. Due to the presence of a pressure gradient within the fluid, after being compressed at the throat and expansion section, the shape of the bubble becomes flat, and finally, it collapses when the external pressure reaches the critical pressure for bubble collapse; The second mode is collapse when a bubble adheres to a solid wall. During the adhesion process, the side of the bubble not in contact with the solid wall is subjected to pressure from the fluid. Under this pressure, the middle area of the bubble gradually sinks and is finally penetrated by the fluid; The third mode is a situation between the first and second collapse forms, where the bubble does not adhere to the solid surface but is close to it and then collapses.

2.2. Governing equations

(1) Mass conservation equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0$$

Where: ρ Denotes the fluid density, t Denotes time, u_i Indicates the velocity of the fluid in the i The velocity component of the direction.

(2) Momentum conservation equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i$$

Where: p Denotes static pressure, τ_{ij} Denoting the stress tensor, g_i , F_i Separate table i Gravitational and external body forces in the direction.

(3) Continuity equation

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0$$

Where: $\bar{u}_i (i = x, y, z)$ means x, y, z Time-averaged velocity components in three directions.

(4) Navier-Stokes equation

$$\bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial \bar{u}_i}{\partial x_j} - \overline{u_i' u_j'} \right)$$

Where: $\bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j}$ Representing the acceleration of a fluid element over space, $\frac{\partial \bar{p}}{\partial x_i}$ Represents the pressure gradient of a fluid cell in different directions. $\frac{\partial}{\partial x_j} \left(\nu \frac{\partial \bar{u}_i}{\partial x_j} \right)$ Represents the force of molecular viscosity, $-\overline{u_i' u_j'}$ Is the Reynolds stress. In the Cartesian coordinate system, the Reynolds-averaged Navier-Stokes equation can be written as follows:

$$\frac{\partial Q}{\partial t} + \frac{\partial F_{inv}}{\partial x} + \frac{\partial G_{inv}}{\partial y} + \frac{\partial H_{inv}}{\partial z} = S + \frac{\partial F_{is}}{\partial x} + \frac{\partial G_{is}}{\partial y} + \frac{\partial H_{is}}{\partial z}$$

The independent variable vector of the equation is,

$$Q = [\rho, \rho v_x, \rho v_y, \rho v_z, \rho E]^T$$

Among v_x, v_y, v_z Is the absolute airflow velocity in the x, y, z The three directional components, ρ Is the density of the gas, E is the total internal energy of the gas, and S is the source term.

Based on the ideal gas assumption, the pressure of the gas is calculated as follows:

$$p = (\gamma - 1)\rho \left[E - \frac{1}{2}(v_x^2 + v_y^2 + v_z^2) \right]$$

The inviscid flux terms are defined as:

$$F_{inv} = \begin{bmatrix} \rho v_x \\ \rho v_x^2 + p \\ \rho v_x v_y \\ \rho v_x v_z \\ \rho v_x H_t \end{bmatrix} \quad G_{inv} = \begin{bmatrix} \rho v_y \\ \rho v_x v_y \\ \rho v_y^2 + p \\ \rho v_y v_z \\ \rho v_y H_t \end{bmatrix} \quad H_{inv} = \begin{bmatrix} \rho v_z \\ \rho v_x v_z \\ \rho v_y v_z \\ \rho v_z^2 + p \\ \rho v_z H_t \end{bmatrix}$$

Where the total enthalpy H_t Is related to the total internal energy, and its calculation formula is,

$$H_t = E + \frac{p}{\rho}$$

The definitions of the viscous flux terms are, respectively,

$$\mathbf{F}_{vis} = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \\ q_x \end{bmatrix} \quad \mathbf{G}_{vis} = \begin{bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \\ q_y \end{bmatrix} \quad \mathbf{H}_{vis} = \begin{bmatrix} 0 \\ \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \\ q_z \end{bmatrix}$$

According to the constitutive equation of Newtonian fluid, the components of viscous stress tensor and heat flux are calculated as follows:

$$\tau_{xx} = 2\mu \left(\frac{\partial v_x}{\partial x} \right) + \lambda_v \nabla \cdot \vec{V}$$

$$\tau_{yy} = 2\mu \left(\frac{\partial v_y}{\partial y} \right) + \lambda_v \nabla \cdot \vec{V}$$

$$\tau_{zz} = 2\mu \left(\frac{\partial v_z}{\partial z} \right) + \lambda_v \nabla \cdot \vec{V}$$

$$\tau_{xy} = \mu \left[\left(\frac{\partial v_x}{\partial y} \right) + \left(\frac{\partial v_y}{\partial x} \right) \right]$$

$$\tau_{xz} = \mu \left[\left(\frac{\partial v_z}{\partial x} \right) + \left(\frac{\partial v_x}{\partial z} \right) \right]$$

$$\tau_{yz} = \mu \left[\left(\frac{\partial v_y}{\partial z} \right) + \left(\frac{\partial v_z}{\partial y} \right) \right]$$

$$q_x = v_x \tau_{xx} + v_y \tau_{xy} + v_z \tau_{xz} + k \frac{\partial T}{\partial x}$$

$$q_y = v_x \tau_{yx} + v_y \tau_{yy} + v_z \tau_{yz} + k \frac{\partial T}{\partial y}$$

$$q_z = v_x \tau_{zx} + v_y \tau_{zy} + v_z \tau_{zz} + k \frac{\partial T}{\partial z}$$

Among μ Is the first viscosity coefficient of the fluid, which is composed of the molecular viscosity coefficient and the turbulent viscosity coefficient of the fluid, and K is the thermal conductivity coefficient. λ_v Is the second viscosity coefficient.

2.3. Turbulence model

The Shear Stress Transport (SST) turbulence model is selected because it can well simulate the separated flow under adverse pressure gradient. For general flow problems, the model is used in the near-wall region $k - w$ Model, but in the region far from the wall $k - \varepsilon$ The model can automatically

convert from the low Reynolds number formula to the wall function treatment according to the grid density, which can effectively avoid the reduction of calculation accuracy caused by the application of the low Reynolds number model under the condition of sparse grid. However, for the low Reynolds number separated flow studied in this paper, it is necessary to accurately solve the viscous underlayer flow in the boundary layer, so when using the SST turbulence model, it is necessary to arrange enough nodes in the boundary layer to ensure the accuracy and accuracy of the solution.

SST turbulence model equation with Langtry correction:

$$\frac{D\rho k}{Dt} = P_k PTM - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right]$$

2.4. Multiphase flow model

Cavitation in Venturi tube belongs to multiphase flow, so this paper uses Fluent multiphase flow module to simulate hydrodynamic cavitation. By comparing the Euler-Euler model, the VOF model and the hybrid model, the discrete model (DPM) is finally selected as the multiphase flow simulation model. The discrete phase model is based on the Euler-Lagrange method, which simulates the interaction between particles and fluid by tracking the trajectory of each particle, and can effectively deal with the problem of gas-solid or gas-liquid two-phase flow, and is suitable for industrial scenarios such as cavitation, spraying and aerosol transmission.

DPM defines the initial position and velocity of discrete particles in the fluid field, and uses the Lagrange method to calculate the trajectory of each particle in the flow field. At the same time, the way of interaction between particles and walls can be set by boundary conditions (such as rebound, trapping, etc.). Overall, DPM is an effective means to deal with sparse granular flow and observe the interaction of particles with fluid and wall. It provides a scientific basis for optimizing the industrial design of Venturi tube model by calculating the movement of each particle and its impact on the surrounding environment.

Trajectory equation of discrete phase model particles:

$$m_p \frac{d\vec{u}_p}{dt} = m_p \frac{\vec{u} - \vec{u}_p}{\tau_t} + m_p \frac{\vec{g} - (\rho_p - \rho)}{\rho_p} + \vec{F}$$

Where: m_p Indicating the particle mass, $\vec{u}$ Represents the fluid phase velocity, $\vec{u}_p$ Represents the particle velocity, ρ represents the fluid density, ρ_p represents the particle density, $\vec{F}$ other external forces, $m_p \frac{\vec{u} - \vec{u}_p}{\tau_t}$ Represents the drag force and represents the drop or particle relaxation time.

When using Fluent software to simulate the Venturi tube, because cavitation is a dynamic model, so a dynamic cavitation model is selected in the simulation process. Full cavitation model (Full Cavitation Model) is a model that can be directly called by Fluent software, which is a model to solve the mass transfer of liquid cavitation two-phase flow with full consideration of the coupling of multi-factors such as liquid surface tension and non-condensable gas concentration. The calling method in Fluent is shown in Figure 2. On the main interface of FLUENT software, input solve, then set, and finally expert, and follow the prompts to operate.

```
> solve

/solve> set

/solve/set> expert
Linearized Mass Transfer UDF? [yes] y
use Singhal-et-al cavitation model? [no] yes
use alternate formulation for wall temperatures? [no] yes
Save cell residuals for post-processing? [no] yes
Keep temporary solver memory from being freed? [no] yes
Allow selection of all applicable discretization schemes? [no] yes
```

Figure 2: Calling Diagram of Full Cavitation Model

3. CFD Model Establishment and Computation for Venturi Tube

The application occasions of computational fluid dynamics (CFD) are flexible. Compared with experimental testing, CFD only requires a computer to carry out corresponding numerical simulations for actual problems, greatly reducing research budget and time costs. In today's industrial field, numerical simulation applications can be seen almost everywhere. This article uses CFD software FLUENT to establish flow field changes inside the Venturi tube to obtain its internal flow rules.

3.1. Model Construction

It is well known that one of the factors affecting numerical solutions of internal flow fields in impeller machinery is the quality of grid within the solution domain. With the rapid development of CFD, grid generation technology has received great attention and achieved significant progress. Advanced grid generation technology and high-quality grid topology structures have emerged accordingly. Currently, two commonly used grid generation methods in CFD are: solving partial differential equations to generate grid numerically and algebraic interpolation to generate grid. All commercial CFD software provides powerful grid generation software.

In FLUENT calculations, better grid quality leads to faster convergence speed and more accurate results. Generally speaking, finer grids mean better quality but increasing grid count will further increase numerical computation time cost, which may lead to ignoring some important factors during numerical computation. Considering these factors, it is necessary to choose a reasonable number and regional division of grids. Structured grids are easy to obtain high-quality and orderly node grids and are generally applicable to regular geometric shapes such as cubes and cylinders. Unstructured grids are more suitable for complex geometries and when dividing multiple geometric bodies.

The Venturi tube consists of a contraction section, a throat, and an expansion section. The dimensions and shapes of these three regions vary significantly. The throat region has rapid velocity increases and pressure drops. Therefore, during grid division, it is necessary to refine grids in the throat to accurately capture these fluid characteristics. The grid type adopts structured grids, and named selections are created at key positions such as inlets, outlets, and walls. To better capture fluid characteristics in the intermediate contraction region, edge sizing and face meshing were used to refine these areas. After completing grid division, grid quality was checked to ensure that orthogonality (Orthogonality) and aspect ratio (Aspect Ratio) are within reasonable ranges, especially for grids near the pipe contraction area. Through these methods, regular hexahedral grids were ultimately obtained, as shown in Figure 3. Local areas were also refined to accurately capture key fluid phenomena during simulation.

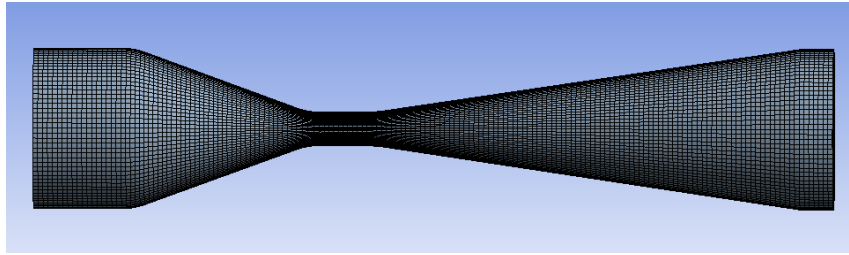


Figure 3: Grid division of Venturi tube cavitation simulation

3.2. Setting Boundary Conditions

Boundary conditions are mathematical and physical conditions that should be satisfied by flow field variables on calculation boundaries. In FLUENT simulation calculations, initial conditions at boundary locations are used for initialization, and iterative calculations are finally performed to solve converged output results. Figures 4. and Figure 5. show the inlet and outlet boundaries set.

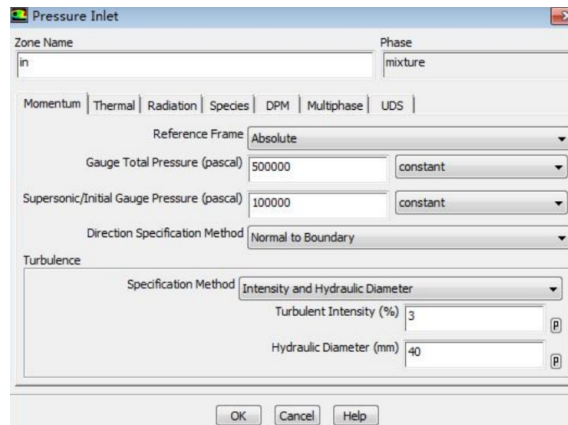


Figure 4: Venturi Inlet Boundary

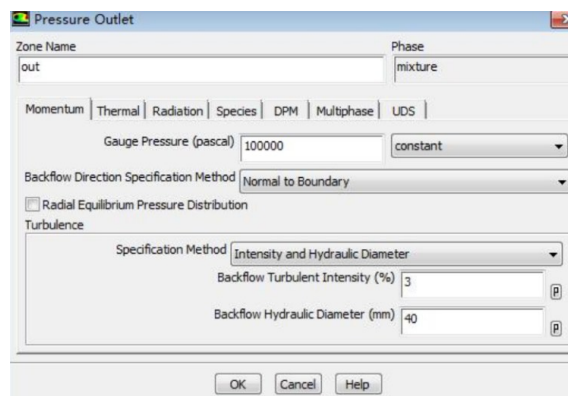


Figure 5: Venturi Outlet Boundary

3.3. Numerical Computations Results

In numerical simulations, comparisons were made between different inlet pressures on Venturi tube flow effects. The results indicate that inlet pressure has a certain impact on internal pressure distribution in Venturi tube. Higher inlet pressure will lead to higher inlet velocities, ultimately making pressure drops more significant in the throat area, increasing pressure differences in the throat region, which may enhance particle separation effects, as shown in Figure 6.

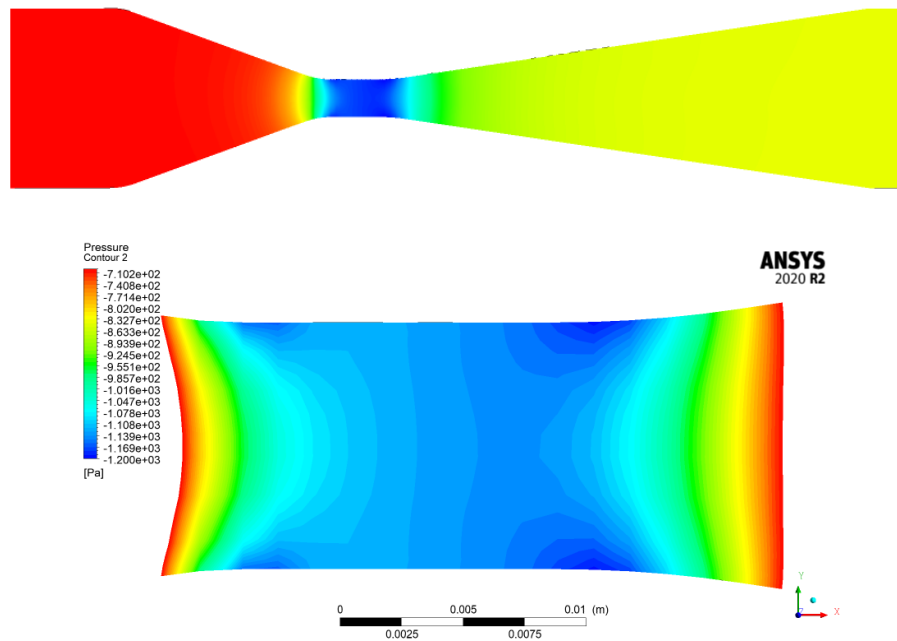


Figure 6: Pressure Distribution of Venturi Tube

4. Experimental Study on Venturi Cavitation Device

A Venturi tube cavitation generator was studied. Visual experiments were conducted using an experimental bench to investigate bubble behavior and gas production characteristics of the Venturi cavitation device.

4.1. Introduction to Experimental System

Figure 7. shows the experimental section of the Venturi cavitation device combined with an air inlet pipe. Due to insufficient preliminary research that did not consider visibility for sampling in the Venturi tube test, it was later replaced with a transparent Venturi pipe as shown in Figure 8..



Figure 7: Experimental device and water circuit



Figure 8: Venturi tube

4.2. Experimental Results

Bubble behavior in the diffusion section of Venturi tube is shown in Figure 9. It can be observed that bubbles generated at the throat are mostly concentrated on the medium surface and are brought into the diffusion section by high-speed shear flow. Bubbles exhibit obvious transverse diffusion in the diffusion section, preventing bubble coalescence. Bubble sizes in the diffusion section are uneven; larger bubbles are further broken down under the action of turbulent diffusion forces.

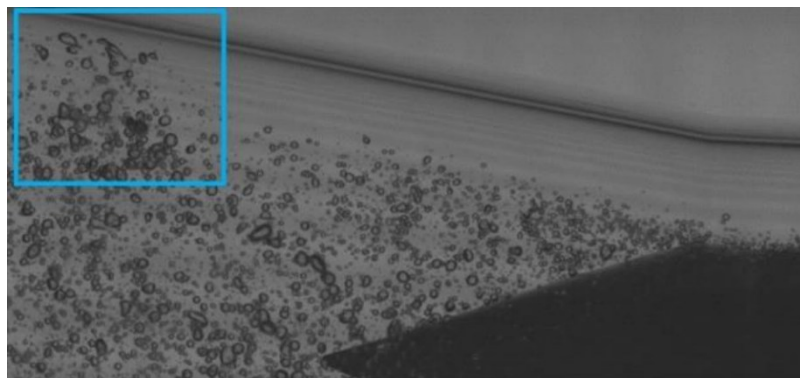


Figure 9: High-speed camera results when cavitation occurs

5. Conclusion

Hydraulic cavitation is an emerging technology with broad application prospects as an efficient and low-energy consumption method. This article takes a Venturi tube hydrodynamic cavitation generator as a research vehicle. Preliminary theoretical analysis was conducted followed by numerical simulation of factors affecting Venturi tube cavitation effects using CFD software FLUENT. Finally, experimental verification was performed to provide guidance for practical engineering applications of Venturi tube hydraulic cavitation. The conclusions are as follows: A dynamic two-phase cavitation model inside Venturi tube was established. This model is based on macrofluid mass, momentum, and energy conservation equations and microscopic individual bubble dynamics model, taking into account shortcomings of current steady-state cavitation models. The establishment of this model provides theoretical support for understanding cavitation evolution processes in Venturi tubes. Based on this model, numerical and experimental studies on a Venturi tube cavitation device were carried out. The results indicate that inlet airflow velocity has a key impact on cavitation effects in Venturi cavitation devices. Due to issues with experimental apparatus preparation, a comprehensive study tailored to practical application situations will be conducted in future work.

References

- [1] Pang, Y., Huang, C., Zhang, Y., & Huang, X. (2020). Refractory DOM in industrial wastewater: Formation and selective oxidation of AOPs. *Chemical Engineering Journal*, 402, 126255. <https://doi.org/10.1016/j.cej.2020.126255>
- [2] Liu Yanan. *Evolution Process and Influencing Factors of Cavitation in Venturi Tube [D]*. Harbin University of Science and Technology, 2019:14-16.
- [3] Dong Zhiyong, Zhang Shaohui, Yang Jie et al. Study on Disinfection Effects of Porous Plate and Venturi Combination Cavitation [J]. *Journal of Zhejiang University of Technology*. 2019(6):6-8.
- [4] Rayleigh L. On the Pressure Developed in a Liquid During the Collapse of a Spherical Cavity [J]. *Philosophical Magazine*, 1991, 34(1):94-98.
- [5] Wang Zhiyong. *Numerical Simulation of Hydraulic Cavitation Based on FLUENT Software [D]*. Dalian University of Technology, 2006