Design of New Combined VENTURI and Hydrodynamic Analysis

Huan Li, Liang GAO*, Zhen-jiang CAI and Xue-song SUO
Mechanical and Electrical Engineering, Agricultural University of Hebei, Baoding, Hebei 071001, China
*Corresponding author

Keywords: Hydrodynamics, Simulation, Fluent, VENTURI.

Abstract. In this paper, aiming at the improvement of VENTURI tube, a combined VENTURI tube is proposed. Using SOILDWORKS for three-dimensional modeling and WORKBENTH for fluid dynamics simulation, the simulation data under different combinations of angles were obtained and the following conclusions were obtained: Under the premise of considering the impact loss, the higher the speed, the greater the negative pressure value. The losses of 45°, 60°, 135°, 150° are respectively 0.059%, 0.064%, 0.12%, 0.27%.

Introduction

Hydrodynamics is an important branch of dynamics. It mainly studies the fluid movement law with water as the main material, including the hydrodynamic load characteristics on moving objects and the pressure field velocity field analysis under various complicated flow conditions. The VENTURI effect, also known as the VENTURI effect, is named after its discoverer, the Italian physicist Vento Giovanni Battista VENTURI. The pipeline due to high flow rate and negative pressure area is widely used, the flow rate and flow cross-section is inversely proportional to the greater the greater the negative pressure flow rate. Bernoulli's law states that the increase of flow rate will be accompanied by the decrease of fluid pressure. Generally speaking, this effect means that low pressure or even negative pressure is generated in the vicinity of the high-speed fluid, thus playing an adsorption role. The use of this effect can produce a venture. VENTURI tubes are used in today's technological developments, benefiting from their lower manufacturing and maintenance costs. In essence, there is a VENTURI mixer that acts as an accurate fertilizer in the greenhouse. For chemical applications there are VENTURI nozzles for removing impurities (removing gas), or for measuring the velocity of a fluid. In recent years, many researchers have used powerful analytical software to conduct dynamic analysis of VENTURI tubes. Among them, the study of cavitation dynamics and hydrodynamic cavitation dynamics occupy most of them and provide guidance in their respective fields sexual improvement. It can be seen; this kind of software analysis of the tube is very necessary.

Theory and Experimental Design

Classic VENTURI tube as shown below, divided into the entrance section, contraction section, throat, diffusion section of several parts.

![Figure 1. Classic Venturi tube chart.](image)
The conservation equations for flow are respectively the continuity equation, the energy equation, the state equation and the momentum equation (where $u$, $v$ and $\omega$ are the velocity components in the $x$, $y$ and $z$ directions, respectively):

1. $\frac{\partial p}{\partial t} + div(\rho u) = 0$  
2. $\frac{\partial (\rho T)}{\partial x} + div(\rho v T) = div \left( \frac{k}{c_p} \text{grad} T \right) + \frac{\rho}{c_p} \frac{\partial T}{\partial T}$
3. $\rho = f(p, T)$
4. $\frac{\partial (\rho u)}{\partial t} + div(\rho u u) = div(\mu \text{grad} u) - \frac{\partial p}{\partial x} + S_u$
5. $\frac{\partial (\rho v)}{\partial t} + div(\rho v v) = div(\mu \text{grad} v) - \frac{\partial p}{\partial y} + S_v$
6. $\frac{\partial (\rho \omega)}{\partial t} + div(\rho \omega \omega) = div(\mu \text{grad} \omega) - \frac{\partial p}{\partial z} + S_\omega$

In this paper, two identical VENTURI tubes were combined at different angles and the combination methods were 45°, 60°, 135° and 150°. The simulation experiments were carried out to observe the formation of water flow trajectory and impact zone. And SOILDWORKS for three-dimensional modeling, as shown below:

![Figure 3. Three-dimensional model.](image)

**Simulation**

In order to predict the feasibility of the design, the design of finite element analysis software ANSYS FLUENT modular VENTURI tube modal analysis. ANSYS FLUENT is a professional fluid mechanics analysis software, with a strong meshing function, rich physical model, advanced numerical methods and powerful pre- and post-processing functions, the following figure is the simulation flow diagram:
Workbench embedded ICEMCFD has a good meshing function, due to the complexity of the model, the design uses unstructured grid computing, by checking the quality of the grid, the grid quality of less than 0.3 area for local encryption processing, delete Unqualified grid. As shown below:

Turbulence model selection, numerical simulation of turbulence in engineering applications can be divided into three types: direct numerical simulation (DNS), large eddy simulation (LES) and Reynolds-averaged N-S equations (RANS). However, due to the constraints of computer conditions, direct simulation method is difficult to deal with complex engineering problems, so the use of N-S equations for solving. The most common method in RANS is the k-ε equation. In FLUENT, the k-ε model is valid both as a low Reynolds number model and as a high Reynolds number model. If you select the Transitional Flows option in the Viscous Model palette, you are using a low Reynolds number variable, so the guidelines for the grid are exactly the same as for the wall function. If you do not check this option, the grid guidelines are the same as for wall functions.

**Post-Treatment Analysis**

The mesh into fluent in the grid to check, you need to ensure that the minimum number of grid greater than or equal to 0. Selecting k-ε model and enabling the effect of gravity, water as an object for numerical simulation, in which the surface tension and density of water, respectively, the default
value. Then defining the combined VENTURI import and exporting conditions. Here double the same import conditions, the export pressure is defined as 0, setting the solution method and controlling parameters.

Set the calculation of the monitor, the import value is the same, choosing an import for monitoring, as shown below (inlet pressure monitoring chart):

![Graph showing import pressure iteration diagram.](image)

Figure 6. Import pressure iteration diagram.

Respectively, the combination of 45°, 60°, 135°, 150° kinetic analysis of these cases, focusing on its pressure field and velocity field comparison, as shown below:

![Graph showing 45° speed pressure cloud chart.](image)

Figure 7. 45° Speed pressure cloud chart.
From the velocity and pressure of the fluid simulation shows that with the increase of angle, speed and pressure in the same order of magnitude increase slightly.

The combination of different angles will form different impact losses, the import and export traffic monitoring, in which the velocity and flow is proportional to the relationship: \( Q \propto V \)

Get various combinations of loss formula: \( \eta = (Q_1 + Q_2 - Q_3)/Q_3 \)

The losses of 45 °, 60 °, 135 °, 150 ° are respectively 0.059%, 0.064%, 0.12%, 0.27%. As the angle increases, the greater the pressure loss caused by the impact of the water flow.

**Conclusion**

Through the dynamic simulation of the combined VENTURI, the following conclusions can be drawn:

1. Modular VENTURI has the similar properties as a classical VENTURI tube. As the flow passes through the reduced tube, the velocity increases and the pressure decreases.

2. The pressure at the corner of the pipe is relatively large and is connected with the negative pressure turbulence zone. Under this condition, the water flows from the two pipes collide with each other, resulting in a greater water flow rate.

3. The combined VENTURI tube tends to increase in confluence velocity and turbulent negative pressure (absolute value) as the combination angle increases.
(4) Combined VENTURI with increasing combination angle, the loss will show an increasing trend.

**Fund support**

This work is supported by the new facilities of agricultural machinery research and development (17227206D), Young Academic Leaders of Hebei Agricultural University

**References**


