Simulation Research on Cavitation Flow Characteristics of Highly Enhanced Diesel Engine Cooling System

ZHANG Bo¹, ZHANG Ping¹, GUO Xu², WANG Yin¹, CHENG Xiao-bei³, HUANG Xin-yuan¹, GONG Xin-rui¹

(¹College of Power Engineering, Naval Univ. of Engineering, Wuhan 430033, China; 
²Navy Representative Office of the PLA in Wuhu, Wuhu, 241000; 
³School of energy and power engineering, Huazhong University of Science and Technology, Wuhan, Hubei 430070)

Introduction to the First Author: ZHANG Bo (1990-), male, Master degree student
Research direction: Monitoring, Control and Fault Diagnosis of Ship Power and Thermal System
E-mail: 821021174@qq.com

Abstract This paper establishes a three-dimensional simulation model of diesel engine single-cylinder cooling water jacket. Using the FLUENT calculation fluid software to extract the boundary conditions of the single cylinder water jacket simulation calculation from the calculation results of the water jacket flow field. Considering the effect of the forced vibration of the cylinder liner wall on the cooling water jacket, the wall vibration UDF function of the cooling water jacket is compiled. Using the dynamic grid technology, the two-phase flow mixing model and the cavitation model proposed by Singhal, the three-dimensional unsteady value simulation of the cooling water in cylinder liner was carried out, and the variation of the pressure and gas content of the single cylinder water jacket was analyzed. The results show that the change period of water jacket pressure and gas content rate is consistent with the diesel engine motion cycle. The simulation results can objectively reflect the characteristics of the cooling water flow field, cavitation and its changing law. Comparing the high gas-bearing area with the actual cavitation position, it indicates that the accumulation of a large number of tiny bubbles easily leads to the occurrence of cavitation in the cylinder liner, which confirms the effectiveness of the simulation method.

1. Introduction
The cavitation of the cylinder liner and the body is one of the main forms of failure [1]. With the increase of diesel engine power and high speed and light weight, the cavitation of the cylinder liner and the body becomes more and more serious [2]. In foreign countries, many scholars have put forward the theory of cylinder liner cavitation and gave analysis methods and preventive measures. Zhou, yu-kang proposed that the cylinder liner damage is mainly the cavitation damage of the cylinder liner. That is to say, the impact caused by the bubble burst has the greatest impact on the cylinder liner damage. In 1998, Japanese scholars started the vibration of diesel engines and analyzed the mechanism of cavitation. They gave formulas for calculating the pressure change of cooling water and the vibration speed of cylinder liner, and made finite element analysis of cylinder liner and piston.
Katraqadda Sunil explained the effect of cavitation on the cylinder liner of diesel engine, analyzed the formation of cavitation, the process of growth and final rupture, and the effects of pressure, temperature and soluble gas on the bubble, and listed the types of damage to the cylinder liner of the diesel engine.

In China, the current research focuses on the causes of cavitation erosion and the prevention and control of cavitation. Ru Luojing et al. calculated and analyzed the mechanism of cavitation morphology formation. It is concluded that the cavitation pit on the surface of the test piece is the result of the combined effects of jet and impact strength and local material strength. Hu Yingying et al. numerically simulates the cavitation damage caused by the collapse of the cavity when the bubble is at different positions of the solid wall. Liu Yinglin and others used ANSYS software to establish a solid model of the 4105 diesel engine block, cooling water and cylinder liner. According to the basic mathematical model of the tribological and dynamic coupling problem of the cylinder liner-piston system of multi-cylinder internal combustion engine, the program for calculating the oil film pressure and friction force is programmed to realize the dynamic coupling in the true sense. Therefore, this paper uses FLUENT calculation fluid software to consider the forced vibration response of cylinder liner as the impact load distributed along the cosine law along the circumferential direction of the cylinder liner, and defines it as UDF function. Using dynamic grid technology to reproduce the flow characteristics and cavitation phenomena of three-dimensional cooling water flow. Using the two-phase flow mixing model and the cavitation model proposed by Singhal, the unsteady numerical simulation of the pressure and void fraction in the flow field was carried out, and the formation, development and fracture process of cooling water cavitation were analyzed.

2. Cavitation flow theory
At present, most of the cavitation models are based on the cavity growth equation [3-5] proposed by Rayleigh-Plesset. The specific form of R-P model equation is as follows:

\[
\frac{d^2 R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 + \frac{4\nu \rho_l R}{R^4} \frac{dR}{dt} = \frac{2S}{\rho_l R} (1)
\]

Where \( P_b(t) \) is the pressure inside the bubble (assuming saturated vapor pressure at liquid temperature), \( P_e(t) \) is the external pressure of bubbles, \( \rho_l \) is a liquid pressure, \( R \) is the radius of cavitation, \( \nu \) is the viscosity of liquid motion, \( S \) is the surface tension of bubbles.

The cavitation of liquid under low pressure is a complex phase transition process. In 2002, Singhal et al. proposed a complete cavitation model considering the effects of various factors, including phase transition, cavitation dynamics and non-condensable gases. In the complete cavitation model, the mixed phase fluid is composed of water, steam and non-condensable gases. The density of mixed phase fluids is defined as:

\[
\frac{1}{\rho} = \frac{w_e}{\rho_e} + \frac{w_g}{\rho_g} + \frac{1-w_e-w_g}{\rho_l} (2)
\]

Where \( \rho_l \) is the density of water, \( w_e \) is the mass fraction of the water vapor phase, \( \rho_e \) is the density of water vapor, \( w_g \) is the mass fraction of non-condensable gas, \( \rho_g \) is the density of non-condensable gases.

The mass fraction transport equation for the water vapor phase is:

\[
\frac{\bar{\partial}(\rho w_e)}{\bar{\partial}t} + \frac{\bar{\partial}}{\bar{\partial}X_j}(\rho w_e u_j) = R_e - R_c (3)
\]

Where \( R_e \) and \( R_c \) are the evaporation rate and the condensation rate.

For the Singhal complete cavitation model, the evaporation rate \( R_e \) and condensation rate \( R_c \) are derived from the R-P equation. The evaporation rate \( R_e \) and condensation rate \( R_c \) are expressed as follows:
Where $k$ is the local turbulent energy of the flow field. $\sigma$ is the liquid surface tension coefficient. Empirical constant $C_c = 0.02, C_e = 0.01$.

Turbulence plays an important role in cavitation. Increasing the initial pressure of phase change should be considered from partial pressure $P_{sat}$:

$$P_{sat} = P_{sat} + P_{turb}$$

Where $P_{sat}$ is the saturated critical vapor pressure of water. $P_{turb}$ is turbulent pressure pulsation. And its calculation formula is:

$$P_{turb} = 0.39 \rho k / 2$$

In this paper, the Mixture hybrid model is selected in the multiphase flow model. This model allows interphase insertion between phases, which is more common when simulating bubble flow. This model allows each phase to be interspersed, and is more commonly used to simulate bubble flow. Gas volume fraction (gas content) is an important physical parameter for characterizing bubble flow. Cavitation model introduces density of mixed phase fluid in calculation. The density of mixed phase fluids can be calculated by the mass fraction of each phase. Combining with the current pressure and temperature of each phase density parameters, the gas mass fraction can be converted into the gas volume fraction, thus realizing the use of gas holdup to characterize the change of bubble flow. In this paper, gas holdup is one of the important parameters to analyze the change of cooling water.

3. **Computational model**

The three-dimensional model of a single-cylinder water jacket is established by using proe three-dimensional modeling software. The main structure distribution and flow channel shape are shown in Figure 1. For the structural analysis of the single cylinder jacket, the water jacket has 3 entries. The front single-cylinder water jacket connects the water cavity inlet, the front channel inlet and the W-shaped water cavity inlet, as shown in the figure Inlet7, Inlet4 and Inlet5 respectively. The water jacket has 6 outlets. On the back side, there are Single cylinder water jacket connected water outlet, bottom connected outlet and 4 outlets for upper water holes, which are shown in the figure Outlet6, Outlet3, and the upper water outlet Outlet1, Outlet2 and Outlet8. Using ICEM CFD software [6-7], the number of grids is 4.23 million, and Figure 2 is a single cylinder water jacket grid.

![Fig. 1 3D model of A2 cylinder single cylinder water jacket](image)
4. Boundary conditions

The water flow field of diesel engine is simulated under the rated condition, which is used as the simulated reference condition, and the wall is set as the fluid without relative slip with the wall. The standard wall function is selected and the RNG equation is used as the turbulence model. A second-order upwind control equation discrete method is used. The energy equation is checked in the software settings, so that the temperature setting information is included in the simulation calculation, and the temperature of the wall and the inlet and outlet is given in segments. The specific parameter settings are shown in Table 1.

| boundary condition                              | set value | Remarks         |
|-------------------------------------------------|-----------|-----------------|
| Inlet velocity                                  | 5m/s      | Bench test      |
| Inlet temperature                               | 342.15K   | Bench test      |
| Upper water outlet temperature                  | 350.15K   | Simulation      |
| Upper water outlet temperature                  | 335 kPa   | Simulation      |
| Supercharger side outlet pressure               | 300 kPa   | Bench test      |
| Supercharger side outlet temperature            | 345.15K   | Simulation      |
| Upper wall temperature of cylinder wall         | 393.15K   | Experience value|
| Temperature at the bottom of cylinder wall      | 373.15K   | Experience value|
| Cylinder wall temperature                       | 343.15K   | Experience value|
| Other wall temperature                          | 343.15K   | Experience value|

The water flow field of a single-cylinder water jacket is simulated under standard conditions. In the calculation of single-cylinder water jacket, cavitation flow simulation method is adopted, and multi-phase flow mixture model is used to extract the boundary conditions of single-cylinder water jacket simulation calculation from the calculation results of the whole machine water jacket flow field. The boundary conditions for import and export are shown in Table 2.

| Import / export | Pressure (Pa) | Velocity (m/s) |
|-----------------|--------------|----------------|
| 1               | 319 554.6780 | 0.163 787 813 |
| 2               | 319 515.6173 | 0.241 028 214 |
| 3               | 319 620.0597 | 0.281 199 398 |
| 4               | 319 689.5872 | 0.277 492 453 |
| 5               | 319 688.3655 | 0.121 528 571 |
| 6               | 319 625.5068 | 0.095 497 780 |
| 7               | 319 622.3432 | 0.443 001 364 |
| 8               | 300 000.0000 |                |
5. Numerical calculation and result analysis

5.1 Law of the change of water jacket pressure and gas content

Fig. 3 is a graph showing the pressure variation curve of a single-cylinder water jacket under reference conditions, and Fig. 4 is a graph showing the gas content [8-9] under a single-cylinder water jacket reference condition. In the interval of 0°~180° of the crank angle in Fig. 3, due to the poor compressibility of the cooling water, in the initial stage of the simulation process, the initial stage of the water flow field has an oscillation interval with the crank angle, however, as the process of simulation calculation advances, the pressure fluctuation will exhibit a periodic variation in the range of 360° to 1080° crank angle.

![Figure 3 the pressure variation curve under reference conditions](image)

![Figure 4 Curve of the gas content of the reference condition](image)

From the variation law of the physical field, the pressure change curve is analyzed: during the change of the crank angle 360°~540°, the wall surface of the cylinder liner is in the rebound phase after the pressure, and the elastic recovery pressure of the flow field fluid with the boundary decreases. A large number of bubbles are generated in the flow field due to the pressure below the critical value. A large number of tiny bubbles rupture near the 540° position of the crankshaft angle, and a micro-jet is generated to impact the metal wall surface. In the pressure curve, a short-time high-pressure impact occurs in the pressure. The sharp peaks are the result of the high pressure of the microjet. During the change of the crank angle 540°~720°, the flow field will increase due to the compression of the cylinder wall surface.

From the variation law of the physical field, the curve of the gas content rate is analyzed: during the change of the crank angle 360°~540°, the wall surface of the cylinder liner is pressed and rebounded, so that the space flow field is poured into the cooling water, and the gas content begins to
decrease. Near the 540° position of the crank angle, the gas content is minimized at this time due to a large number of bubbles rupturing. During the change of the crank angle 540° ~ 720°, the flow field is squeezed by the wall surface of the cylinder liner, the cooling water is squeezed out of the space flow field, and the near wall bubble has the opportunity to contact the metal wall surface, which is reflected in the upward trend of the gas content curve.

The curve change period is consistent with the diesel engine's operating cycle, and the fluctuation amplitude is also within a reasonable range. The simulation results can objectively reflect the pressure and gas content rate changes in the flow field.

5.2 Simulation calculation of key distribution area of gas content
The simulation calculation was performed on the single-cylinder water jacket, and the results are shown in Fig. 5. Taking the gas content as the reference physical quantity, focusing on the part where the single-cylinder water jacket is prone to gas-liquid phase change [10], it can be visually observed that the highlighted color display area in the figure is a region with a high gas content rate, and its gas content rate range. It is about 0.34% - 0.94%. This area is the key area for the generation and collapse of microbubble flow. It is the key area for analyzing the damage generation mechanism and comparing the actual damage occurrence position.

Figure 5 Statistical range of gas content and pressure curve
According to Fig. 5, comparing the actual cylinder cavitation position: the simulated key area accurately covers the damage area of the outer wall surface of the cylinder liner, but only expands outward at the edge position. The reason is analyzed: the simulated gas-bearing rate from the center of the key area to the edge of the key area is decreasing, that is, the gas content of the edge of the key area is low, which does not necessarily cause extremely bad influence on the effect of the metal wall. Therefore, there is a transition area from the edge to the central area, which is consistent with the actual observation of the photo of the cylinder liner. It is proved that the location of the key area simulated by the single-cylinder water jacket coincides with the position of the cylinder liner of the actual machine, which confirms the effectiveness of the simulation method.

5.3 Change of pressure and gas content in the flow field of key areas
Through the previous analysis, it can be seen that the key area is a single-cylinder water jacket away from the W-shaped water chamber side, and the key areas are longitudinally observed to understand the distribution of the single-cylinder water jacket flow field in the critical area. The observation section can be taken from the center of 0 mm, ±10 mm, ±20 mm, and ±50 mm along the x-axis direction. Figure 6 is a schematic view of the longitudinal cutting of the key area. Figure 7 is a schematic cross-sectional view of the observation. Fig. 8 is a comparison diagram of each observation section. The variation of each section has a similar distribution law. The observation section at the center of 0 mm is representative, and the pressure and gas content inside the flow field in the key area are analyzed with time. FIG. 9 is a schematic cross-sectional view of a central position, and 10 is a
schematic view of a cloud at a central position.

Figure 6 Schematic diagram of vertical cutting of key areas

Figure 7 Schematic diagram of the cross section

Figure 8 Comparison of each observation section

Figure 9 Schematic diagram of the center position observation section

The pressure distribution and the gas content distribution in the three vibration cycles were observed and recorded with the crank angle as the cycle change reference and the crank angle of 180° as the observation interval. Table 3 gives the distribution of the single-cylinder water jacket pressure
and gas content change distribution in the baseline conditions.

![Schematic diagram of the center position cloud](image)

**Table 3 Distribution of single-cylinder water jacket pressure and gas content change in the reference condition**

| Crank angle/°CA | Pressure distribution cloud map | Gas rate distribution cloud map |
|-----------------|---------------------------------|---------------------------------|
| 180             | ![Pressure cloud chart (180°)]  | ![Gas ratio nephogram (180°)]   |
| 360             | ![Pressure cloud chart (360°)]  | ![Gas ratio nephogram (360°)]   |
| 540             | ![Pressure cloud chart (540°)]  | ![Gas ratio nephogram (540°)]   |
Table 3 shows the distribution of the key pressure and gas content change of the single-cylinder water jacket as the crank angle changes. It can be seen from the figure that the gas-liquid phase change of the cooling water in the critical area of the single-cylinder water jacket, the gas-liquid phase of the fluid is an important source of a large number of tiny bubbles in the flow field, and the accumulation of a large number of tiny bubbles easily leads to the occurrence of cavitation in the cylinder liner. Therefore, the key area with high gas content in the water flow field should be an important part of our concern for the damage of the cylinder liner.

6. Conclusion
In this paper, a high-enhanced diesel engine single-cylinder water jacket is used as the research object under standard working conditions. Based on the three-dimensional CFD model, the two-phase mixed flow model and the cavitation model proposed by Singhal are used to simulate the cavitation dynamics of the cooling water flow field pressure and gas content, and the cavitation process of the liquid flow field is analyzed. Verify that the cylinder liner is the most prone to cavitation, and draw the following conclusions:

(1) The water jacket pressure and gas content curve and the change period are consistent with the diesel engine operating cycle, and the fluctuation amplitude is also within a reasonable range. The simulation method can objectively reflect the pressure and gas content change law in the flow field;

(2) The simulated key area with high gas content rate coincides with the position of the cylinder liner of the actual machine, confirming the effectiveness of the simulation method;
(3) The gas-liquid phase change of the fluid in the key area of the water jacket is an important source of a large number of tiny bubbles in the flow field, and the accumulation of a large number of tiny bubbles easily leads to the occurrence of cavitation in the cylinder liner; 
(4) Based on the CFD simulation method, the influence of gas-liquid phase change on the gas pocketping rate on the cavitation of the cylinder liner was analyzed, which provided a technical platform for analyzing the mechanism of crust erosion and improving the cavitation erosion.

Acknowledgment
Fund projects: Pre-research project of the “13th Five-Year Plan”(3020401030403); Hubei Natural Science Foundation Project (2016CFB62)

Reference
[1] Kai Yuan Tang, Guang-Yao Ouyang. Advanced Internal Combustion Engine [M]. Beijing: National Defense Industry Press, 2008.
[2] Guang-Yao Ouyang, Chang Han Bao. Internal Combustion Engine [M]. Beijing: National Defense Industry Publication, 2011.
[3] Xia Dong Sheng, Zhang Hui Chen, Zhang Xin Wei. CFD-based numerical analysis of the cavitation characteristics of cooling water in cylinder liner of diesel engine [J]. Journal of Internal Combustion Engine, 2010 (4): 368-373.
[4] Makkapati S, Poe S, Shaikh Z, et al. Coolant Velocity Correlations in an IC Engine Coolant Jacket[R]. SAE Technical Paper, 2002.
[5] Porot P A, Ménégazzi P, Ap N S. Understanding and Improving Evaporative Engine Cooling at High Load, High Speed by Engine Tests and 3D Calculations[R]. SAE Technical Paper, 1997.
[6] Zhang Kai. Fluent Technology Foundation and Application Example [M]. Beijing: Tsinghua University Press, 2010.
[7] Franz J Laimböck, Gerhard Meister, Simon Grilc. CFD Application in Compact Engine Development[R]. SAE Technical Paper, 1998.
[8] Liu Qi. Study on Boiling Heat Transfer and Two-Phase Flow in Cylinder Head Cooling Water Cavity [D]. Wuhan: Naval Engineering University, 2015.
[9] Chen Jian. Cavitation mechanism of diesel engine cylinder liner [J].Journal of Shanghai Normal University (Natural Science Edition), 2000, 29 (1): 63-69.
[10] Gao Han. Simulation and Analysis of High Power Density Engine Cooling System [D]. Beijing Jiaotong University, 2007.