Development of gaseous cavitation model in hydraulic oil flow considering the effect of dynamic stimulation

Y Ibata¹, Y Matsuura¹, S Takahashi¹, and Y Iga²
¹Graduate School of Engineering, Tohoku University, Sendai, Japan
²Institute of Fluid Science, Tohoku University, Sendai, Japan
E-mail: ibata@cfs.ifs.tohoku.ac.jp

Abstract. Hydraulic equipment is widely used in the industrial field. For example, driving equipment such as power steering of automobiles, and construction machinery such as hydraulic excavators, and so on. The oil used in these machines is called hydraulic oil. In hydraulic oil system, the cavitation has an influence on its durability and efficiency, therefore the prediction of cavitation phenomena has an important meaning for the design of hydraulic equipment. However, experimental prediction is difficult because hydraulic equipment is used under severe conditions of high pressure. Generally, cavitation is classified in two types. One is caused by evaporation of working fluid which occurs when the local static pressure goes lower than the local saturated vapor pressure, called vaporous cavitation. The other is by the liberation of dissolved air, called gaseous cavitation. Among these two types, the vaporous cavitation is considered predominant in the water. On the other hand, in hydraulic oil, the saturated vapor pressure in the standard state is usually assumed to be close to 0 Pa. Therefore, in hydraulic oil, gaseous cavitation due to air dissolved in oil at a relatively high volume ratio is considered to be predominant and the mechanism of occurrence, which is completely different from cavitation occurrence with water. In the past researches, in order to understand the characteristics of gaseous cavitation, a number of studies has been conducted. However, regarding a specific cause of the liberation of dissolved air, the threshold values and formulas that serve as criteria are not proved. In this paper, the cavitation phenomena in hydraulic oil is investigated, and a new gaseous cavitation model based on the dynamic stimulation is presented.

1. Introduction

In these days, variety of oils has been used in industrial scenes. They are classified and modified to meet various kinds of our needs. Among those oils, hydraulic oil is widely used to the power transmission, ranging from the construction equipment to the brake system of motorbike. In the hydraulic oil system, a phenomena called cavitation can be occurred, and it is known that this phenomena has an influence on decline of its durability and efficiency. Therefore the prediction of cavitation phenomena has an important meaning for the design of hydraulic equipment. However, because hydraulic equipment is used under severe conditions of high pressure, experimental prediction is hard to be conducted. This is why the numerical analysis of oil flow is now on demand.

Cavitation is distinguished in two types. One is caused by the phase change of working fluid which occurs when the local static pressure goes down about the local saturated vapor pressure, called "vaporous cavitation". The other is by the liberation of dissolved air, called "gaseous cavitation". Since the hydraulic oil generally contains 8.0 - 10.0 Vol.% of air, which is much larger than that in water (2.0 Vol.%)[1], gaseous cavitation is supposed to be predominant in the hydraulic oil flow. In most of existing cavitation models of CFD, vaporous cavitation is considered as a predominant factor and there are only a few models
taking gaseous cavitation into account. When considering hydraulic oil flow, the numerical flow analysis based on vaporous cavitation should not prove its reliability, due to its cavitation formation. Therefore, the new cavitation model for the gaseous cavitation, which means considering liberation of dissolved gas, must be developed.

To understand the occurrence characteristics and the dominant factor of gaseous cavitation, a number of experiment has been conducted in past researches using hydraulic oil. From those studies, it is suggested that dynamic stimulation by the flow may influence the liberation of dissolved air. However, studies have not yet clarified what the specific cause of the liberation is, and accordingly, they have not found neither the threshold values nor formulas for liberation. Overall, problems of CFD in hydraulic oil flow is that, there is no certain cavitation model which takes gas cavitation as predominant, and which considers the effect of dynamic stimulation by flow field. Therefore the aim of this study is to develop a new cavitation model which can simulate gaseous cavitation, by considering the effect of dynamic stimulation. To achieve this goal, firstly the cavitation phenomena in hydraulic oil is investigated by the experimental approach, and then a new gaseous cavitation model based on the dynamic stimulation is presented and validated to be able to simulate the cavitation phenomena in oil flow macroscopically.

2. Preliminary Experiment

2.1. Experimental setup

In this study, the flow path is used the experimental apparatus constructed in the previous research[2]. At first, preliminary experiment is done in order to show the state of cavitation in hydraulic oil. The circuit diagram of the experimental apparatus is shown in Figure 1, and the cavitation occurrence in the test section of experimental apparatus is shown in Figure 2. There is triangle convering and sudden divering nozzle in the test section. The test section height is 12 mm, and the throat span height is 1 mm. The test section width is 3 mm, the plate width is as thin as 3 mm, and therefore it is regarded as a 2D flow path. In this flow field, except for the throat section, it is under high pressure or atmospheric pressure, and it is unlikely that the liberation occurs except in the throat section. This experiment is conducted under the conditions shown in the table1. As a test hydraulic oil in this study, Daphne Hydraulic Fluid46 by IDEMITSU KOSAN Co., Ltd. has been selected.

2.2. Experimental result

The visualization result is shown in Figure 3 and Figure 4. These are the state of cavitation occurring with the acceleration of the flow. 0 ms is the time when flow generation was visually confirmed. From
Table 1. Experimental conditions.

| Temperature $\theta$ [°C] | Inlet Pressure $p_{in}$ [MPa] | Outlet Pressure $p_{out}$ [MPa] | Flow Velocity $u_{in}$ [m/s] |
|---------------------------|-------------------------------|-------------------------------|-----------------------------|
| 20                        | 2.19                          | 0.13                          | 3.45                        |
| 40                        | 0.90                          | 0.12                          | 2.21                        |

![Streamline and cavitation generation](image1)

**Figure 3.** Visualization of streamline and cavitation generation in Daphne Hydraulic Fluid46. ($\theta = 20 ^{\circ}$C, $u_{in} = 3.45$ m/s, shot speed 30,000 fps, shatter speed 1/900,000sec)

![Streamline and cavitation generation](image2)

**Figure 4.** Visualization of streamline and cavitation generation in Daphne Hydraulic Fluid46. ($\theta = 40 ^{\circ}$C, $u_{in} = 2.21$ m/s, shot speed 30,000 fps, shatter speed 1/900,000sec)

these figure, a thing like "line" (the red circle) was observed after the flow started. It is thought that this is not a cavitation but a separation streamline by Washio et al’s research[3]. It can be seen that bubbles are generated at a position away from the throat section, regardless of whether the oil temperature is 20 or 40 °C. Therefore, in development and the validation of the gaseous cavitation model from the next section, it is assumed that the position away from the throat is set as the cavitation inception point, the cavity develops in the downstream area.

### 3. Numerical Method

#### 3.1. Gaseous cavitation model

When considering the gaseous cavitation occurrence, the air separation pressure is generally taken as the threshold pressure. However, it is known that the value of air separation pressure changes according to the flow field and has not been clarified. Therefore, in this study, the air separation pressure is determined considering the macroscopic dynamic stimulation accompanied by the unsteady flow with cavitation. The dynamic stimulation by the velocity is transferred into the dimension of pressure and applied to the air separation pressure $p_s$. Then the separation pressure is determined by the variable and local value. Considering the liberation occurs when the local static pressure takes lower value than the air separation pressure, the liberation speed $\dot{m}$ is expressed as follows[4]:

$$\dot{m} = C_m \alpha (1 - \alpha) (p_s - p)$$  (1)

Here, $C_m$ is empirical liberation constant. The function $\alpha (1 - \alpha)$ makes $\dot{m}$ zero at $\alpha = 0$ and 1, where there is gas-liquid no interface. Although, $\dot{m}$ should be zero when the dissolved air is all liberated or is saturated in oil regardless of void fraction $\alpha$ in the viewpoint of actual phenomenon, but the mechanism
is not taken into account in this study. In addition, solution of air proceeds when the $p$ shows higher value than $p_s$. By replacing this liberation speed $\dot{m}$ with the evaporation model in the source term, this gaseous model is applied to the commercial CFD software.

The proper element of the dynamic stimulation inducing gaseous cavitation has not yet been observed neither experimentally nor numerically, and furthermore, then the reliable value of air separation pressure has not yet been defined. Therefore we assume various kind of dynamic stimulation and examine, which are shown as follows:

Model 1

$$p_s = C_s \rho \left| \overline{u'_i u'_j} \right|,$$

(Reynolds stress) (2)

Model 2

$$p_s = C_s \rho \frac{L}{u_w} \left| \overline{u'_i u'_j \frac{\partial \overline{u_i}}{\partial x_j}} \right|,$$

(Source term of turbulent energy transport equation) (3)

Model 3

$$p_s = C_s \rho_0 L^2 \max(0, -2 \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i}),$$

(2nd invariant of velocity tensor (Q criteria)) (4)

Model 4

$$p_s = C_s \rho_0 L u_in \left| \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right|.$$

(Rate of strain) (5)

Here, $C_s$ is empirical liberation model constant, $L$ is the characteristic length. Also, $u'_i, u'_j$ of model 1 and 2 are the local fluctuation components of velocity, and the $u'_i, u'_j$ value of the grid scale obtained from the velocity distribution of the calculation result is used. Each model contains the dynamic stimulation by Reynolds stress, source term of turbulent energy transport equation, 2nd invariant of velocity tensor (Q criteria), and rate of strain, respectively. In this study, we evaluate macroscopically these models and examine which model match macro-scale cavitation distribution in the actual flow field.

3.2. Computational code

In this study, a commercial CFD software is used. For the software to be used, it is possible to flexibly customize the calculation method, it is expected to be highly reliable as a solver, and it is widely used for numerical analysis of the hydraulic equipment and it is easy to apply to industrial applications[5][6], ANSYS Fluent manufactured by ANSYS Co., Ltd. was used. Using the User Defined Function(UDF) for ANSYS, the gaseous cavitation model proposed in this study is applied to the phase change term $\dot{m}$. Table 2 shows the settings of numerical method and so on. In Fluent, cavitation flow field is analysed by incompressible method.

3.3. Computational condition

For the analysis condition, the experimental condition at $\theta = 40^\circ C$ in the section 2 is referred to: the pressure at upstream of the throat $p_{in} = 0.90$ MPa, that at downstream $p_{out} = 0.12$ MPa, inlet velocity $U_{in} = 2.21$ m/s. The boundary condition is set as follows: inlet void fraction, inlet velocity and outlet pressure is set as constant. The other properties (inlet pressure, outlet velocity) are given by 0th order extrapolation. Wall boundary is non-slip condition. The calculation area is a 2D throat flow path whose width of the throat section is 1/12 of the flow path width as shown in Figure 5. ANSYS Meshing is used for grid generation. The number of elements is 216,000 with a structural grid, and the maximum aspect ratio is about 550.
Table 2. Main properties in ANSYS Fluent.

| Genus       | Setting                  | Optional settings                      |
|-------------|--------------------------|-----------------------------------------|
| Solver      | Pressure-based           | Slip velocity OFF                       |
|             | Transient                | Low-Re correction                       |
|             |                          | Production Kato-launder                 |
|             |                          | Production limiter (set by UDF)         |
| Model       | Multiphase Mixture       | Phase-interaction 0 (set by UDF)        |
|             | Viscous SST k – \omega  |                                        |
| Material    | Primary Oil (Liquid)     | Oil (Property referred from Hydraulic Fluid 46) |
|             | Secondary Air (Gas)      | Air                                     |
| Pressure-velocity coupling | SIMPLE                  |                                        |
| Discretization method | Gradient Least square cell based Wrapped-Face gradient correction |
|             | Pressure Body force weighted |
|             | Momentum First order upwind |
|             | Volume fraction High order term relaxation |
|             | Turbulent Second order upwind |
|             | kinetic energy           |                                        |
|             | Specific Second order upwind |
|             | Dissipation Rate         |                                        |
|             | First order upwind       |                                        |
| Transient formulation | First order implicit    |                                        |
4. Results and Discussion

4.1. Distribution of air separation pressure

In order to develop a gas cavitation model considering dynamic stimulation, comparison between the experiment result and the calculation result of the model shown in equation (2)–(5) is done. Figure 6 shows the comparison results for each air separation pressure $p_s$ evaluated from same flow field calculated without the present gaseous cavitation model. The model constant $C_s$ is set to 1 in all models for just comparison of the distribution. Instantaneous distribution of the air separation pressure is shown in each model. As shown in equation (1), liberation is promoted in the cavitation model when the air separation pressure $p_s$ is higher than the local static pressure $p$. Then this result roughly corresponds to the cavitation occurrence region. Each model is evaluated by how its numerical distribution agrees with our experimental visualization in same flow field.

Computation time is not sufficient for all models, and a large-scale vortex structure due to the initial value remains in the downstream area of the throat (the White circle). However, Model 1 and 2 show a high air separation pressure region in the downstream away from the throat tip, which coincides with the cavitation occurrence region observed in the experiment. On the other hand, in model 3 and 4, a region with a high air separation pressure is distributed at the tip of the throat in common, the result in Model 3 and 4 does not coincide with the actual phenomenon. As a reason for such a difference, it is conceivable that Model 1 and 2 takes the local fluctuation component of the velocity into consideration, whereas Model 3 and 4 takes the absolute value of the velocity into consideration. Therefore, it will be predicted
that the cavitation in the hydraulic oil can be reproduced by considering the local fluctuation component of the velocity.

Next, based on the value of $p_\text{s}$ obtained, the model constant $C_s$ in each model is determined so that $p_\text{s}$ takes a value of the same order in each other and appropriate value for liberation. Table 3 shows the values of $C_s$ used in this study. In the table "Range of values", it shows the applicable range of constants without diverging the calculation. In this study, "Applied Value" which is the most suitable the air separation pressure distribution among them is used. Here, "Applied Value" is determined by heuristic method.

### Table 3. Model constant for gaseous cavitation model.

| Control parameters | No. of Air separation pressure model | 1  | 2  | 3  | 4  |
|-------------------|-------------------------------------|----|----|----|----|
| $C_m$             |                                     | 0.0001 |    |    |    |
| Range of value    |                                     | 10–250 | 0.01–10 | 0.001–0.1 | 0.1–100 |
| Applied value     |                                     | 100 | 5 | 0.1 | 25 |

#### 4.2. Cavitation reproduction by the present gaseous cavitation model

The result of instantaneous distribution of void fraction by applying the cavitation model to the flow field is shown in Figure 7. In Model 2 and 3, the distribution is also shown in which the time marches, because the calculation is stable in Model 2 and 3. Overall, it showed the same tendency as the distribution of $p_\text{s}$. In Model 3(Figure 7(d)), a cavity is occurred linearly from the tip of the throat, and in Model 4(Figure 7(e)), cavitation occurred from the tip of the throat and the top wall. These results are confirmed the difference from the experiment result. On the other hand, although a large vortex structure remains in Model 1(Figure 7(b)) and 2(Figure 7(c)), cavitation is occurred at the slightly downstream part from the throat, and results close to the experiment is obtained. Considering the stability of calculation, With the exception of Model 2 and 3, the calculation time is not sufficient for all models, and a large scale vortex structure caused by the initial value remains in the downstream area of the throat. In addition, the void rate increased due to the influence of the vortex caused by this initial value. As a result, incompressible solution became unstable, calculation other than Model 2 and 3 became unstable and diverged. Therefore, Model 2 can reproduce the occurrence position of cavitation in the experiment relatively well in this flow field, and the calculation is considered to be relatively stable in general purpose code.

From these reasons, in this study, we conclude that Model 2 modeling the source term of turbulent energy transport equation as a dynamic stimulus is optimal as a gaseous cavitation model in this case. That is because it is possible to reproduce the occurrence position of cavitation in the experiment relatively well and the calculation is relatively stable in Fluent. As a future examination subject, in order to obtain reproducibility of the cavity distribution more accurately, it is necessary to investigate analytical methods such as physical property values and compressibility, and investigate the applicability to other flow field such as that with more moderate separation.
Figure 7. Distribution of void fraction by gaseous cavitation model.
5. Conclusion

In this study, the characteristics of cavitation occurrence in the hydraulic oil is investigated, and a new gaseous cavitation model considering the effect of dynamic stimulation is examined. We have obtained those results shown below:

- From the visualization of the hydraulic oil cavitation phenomenon in the 2D throat path, we observed that the point of cavitation occurrence is the position away from the tip of the throat on the downstream side.

- We proposed a new gaseous cavitation model that changes the air separation pressure locally and unsteadily by considering the dynamic stimulation by flow, and we compared it with real phenomenon. As a result, the air separation pressure distribution closest to the cavitation occurrence region in the actual flow path was obtained when the local fluctuation component of the velocity was used as an element for determining the air separation pressure distribution.

- The gaseous cavitation model proposed in this study was applied to ANSYS Fluent and a model suitable for the reproduction of the gaseous cavitation was investigated. As a result, when source term of turbulent energy transport equation is taken into account as a dynamic stimulation, a cavity distribution closest to the occurrence phenomenon of actual cavitation is obtained.

References

[1] Nakagawa K, Nonoshita T, and Takahashi K, 1994, "A Study on the Air Release Model during Oil Column Separation", Hydraulics & Pneumatics., 25(3), May, pp.115–122.

[2] Tateishi S, 2017, "Study on the Mechanism of the Cavitation Generation in Hydraulic Oil in the Two-dimentional Nozzle", Master's Thesis, Tohoku University, Feb.

[3] Washio S, Takahashi S, Uemura K, Iwamoto T, and Ogata T, 2008, "Singular properties of flow separation as a real cause of cavitation inception", Proc. IMechE - Part C: J. Mech. Eng. Sci., 222(4), pp.667–678.

[4] Ochiai N, Iga Y, Nohmi M, and Ikohagi T, 2013, "Study of Quantitative Numerical Prediction of Cavitation Erosion in Cavitating Flow", J. Fluids Eng., 135(1), pp.011302.

[5] Tsukiji T, Takase T, and Noguchi E, 2011, "Visualization Analysis of Cavitation Jet Flow Issuing from Notch in an Axial Piston Pump", Transaction of the Japan Fluid Power System Society, 42(1), pp.7–11.

[6] Jackson R S, Belger M M, Cassaidy K J, and Kokemoor A R, 2017 "CFD Simulation and Experimental Investigation of Steady State Flow Force Reduction in a Hydraulic Spool Valve With Machined Back Angles", ASME/BATH 2017 Symposium on Fluid Power and Motion Control, pp.V001T01A061--V001T01A061