Noise investigation of a cavitating orifice: use of cfd simulation and fflowcs williams–hawkings (fw-h) formulation to gain insight into acoustics condition

A Bashirzadeh Tabrizi¹ and B Wu¹
¹Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, China
bashirzadeh@hotmail.com

Abstract. In piping systems, hydrodynamic cavitation is well known for its destructive capabilities such as generation of intense noise. The investigation of the effect of the cavitation on acoustic pressure fluctuations and resonance frequencies provides much information about the geometry of the cavitation area, the propagation and the damping characteristics of the cavitation medium. The acoustical effects of cavitation caused by a water flow through a single-hole orifice have been investigated by using Computational Fluid Dynamics (CFD) and Ffowcs Williams–Hawkings (FW-H) formulation. The fluid zone downstream of the orifice where the cavitation happens considered as acoustic source. Time-accurate solutions of the flow-field variables are obtained from large eddy simulation (LES). Two cavitation regimes are investigated and for each regime, the sound pressure signals far downstream of the orifices are computed by applying the Ffowcs Williams–Hawkings (FW-H) formulation. From the results, it can be concluded that the spectral characteristics of the sound pressure fluctuations are influences by the cavitation condition. The super cavitation condition generates significantly more noise than the developed cavitation condition. Moreover, the results demonstrate that a combination of LES and FW-H formulation is a promising tool for acoustical study of cavitating orifices.

1. Introduction
Cavitation generally occurs when the pressure in specific locations drops below the vapor pressure in liquid flows, and as a result, cavities which are filled with gas and vapor are formed. In a wide variety of hydraulic phenomena, cavitation can be observed, and the cavitation flow usually causes a lot of unpleasant results[1]. In a valve or orifice, a local increase of flow velocity at the location where the flow area decreases, drops the fluid pressure. When the pressure drops below the saturated vapor pressure, cavitation bubbles appear. Downstream from the throttle, flow velocity decreases and the fluid pressure rises and causes cavitation bubbles to collapse[2]. In industrial phenomena, cavitating flows may generate significant levels of noise and high vibrations of structures[3].

Industrial pipe systems include many types of flow equipment such as valves, taps and orifices. In specific flow regimes, single-hole orifice may experience cavitating flow that generates a high level of noise, resulting in acoustic nuisance to workers in the installations and to people in the vicinity[4]. Hydrodynamic cavitation is well known for its destructive capabilities such as material damage and
generation of intense noise in piping systems, in which cavitation is indicated as a significant source of noise. For these reasons, the priority in system design has been to avoid or control cavitation [5, 6].

Restriction orifices such as single-hole, multi-perforated and cone type orifices are used in chemical, process and power plants to restrict the flow in the piping systems. There are some investigations about the cavitation characteristics on the pipe vibration and noise for the above three types of restriction orifices. Those investigation results can be applied to design the restriction orifices in order to avoid the pipe vibration and noise generated by cavitation[7, 8]. Also, fatigue issues have been reported in a cavitating valve and a cavitating orifice[9, 10].

In this study, the acoustical effects of cavitation caused by water flow through a single-hole orifice have been investigated using the large eddy simulation (LES) and Ffowcs Williams–Hawkins (FW-H) formulation. The fluid zone downstream of the orifice where the cavitation occurs is considered as the acoustic source that generates sound. Time-accurate solutions of the flow-field variables, such as pressure and velocity components on the source surfaces, are obtained from the LES. Two regimes of cavitation are investigated and for each regime, the sound pressure signals further downstream of the orifices are computed by the FW-H formulation.

2. Model geometry and water characteristics
In this study a single-hole orifice investigated experimentally by Testud et al. in terms of noise generated by cavitating flow has been modelled[3]. The flow characteristics of the transient regime downstream of the orifice as the sound source were computed by using CFD and far-field noise predicted based on the FW-H formulation. The geometry characteristics of the orifice are as follows:

It is a single-hole, circular, centered orifice, with right angles and sharp edges. Its thickness (t) is 0.014m and its diameter (d) is 0.022m. The orifice is located inside of a hydraulically smooth steel pipe of 0.074m inner diameter (D) and 0.008m wall thickness (tp). Since \( \frac{t}{d} \leq 2 \), the orifice is defined as a “thin” orifice.

The length of the pipe is included in the CFD calculation domain to find time-accurate solutions of the flow-field variables on source surfaces (fluid zone downstream of the orifices). The pipe length is 5D upstream of the orifices and 15D downstream of the orifices. Figure 1 shows the calculation domain, sound source and sound receiver locations. The water characteristics applied in this study are shown in table 1:

| Parameter                                      | Value          |
|------------------------------------------------|----------------|
| Temperature [K]                                | 310            |
| Dynamic viscosity of water \( (\mu_w(310K)) \)[kgm\(^{-1}\)s\(^{-1}\)] | \( 8.86\times10^{-4} \) |
| Density of water \( (\rho_w(310K)) \)[kgm\(^{-3}\)] | 994            |
| Speed of sound in pure water [ms\(^{-1}\)]       | 1,420          |
| Water vapour pressure \( (P_w(310K)) \)[Pa]       | \( 5.65\times10^{3} \) |
3. Numerical procedures

3.1. Mathematical models, mesh and unsteady simulation parameters

Three-dimensional, Navier-Stokes equations are solved for incompressible flow. The LES approach is applied to simulate the unsteady large scale structures and separation zones. In LES, large eddies are solved directly, and small eddies are simulated with a sub-grid scale model. A main function of the small eddies is to dissipate the turbulent energy which is transferred from the larger scales to the smaller scales by the energy cascade[11]. In the LES method, low-pass spatial filtering of the governing equations is deployed to achieve the separation between the resolved and unresolved scales. The subgrid-scale stresses which result from the filtering operation are uncertain, and need to be modelled[12]. LES is the most appropriate approach for acoustic applications. The approach is a compromise solution between DNS and RANS. All scales are numerically solved in DNS, while in RANS, all scales are modelled. In LES, the transport equations are filtered, only larger eddies are resolved, and the smaller eddies modelled. Therefore, LES is an efficient method of achieving good results in turbulent flows[13]. As LES requires that only larger eddies resolve, coarser mesh and a larger time step can be applied compared to DNS, but still a much finer mesh is needed compared to other turbulent models. LES has to run for a long flow-time to obtain statistics so the flow can be modelled and achieve good results. As a result, computational costs in terms of RAM and CPU are higher than RANS models and high-performance computing is required[14].

In this case, Adapting Local Eddy-Viscosity (WALE) model is used. 0.5 is the published value for the WALE constant(Cw) but validation during a European Union research project has presented consistently superior results in ANSYS Fluent with Cw = 0.325, and therefore this value is used as the default setting. The WALE model is designed to provide the correct wall asymptotic (y\( ^+ \)) behaviour for wall bounded flows. In addition, the WALE model provides a zero turbulent viscosity for laminar shear flows and allows the correct treatment of laminar zones in the domain[11].

Schnerr and Sauer model is utilized to include cavitation effects in two-phase flows while the mixture model is applied[11, 15]. Water-liquid and water-vapour are used as primary phase material and secondary phase material, respectively. The frequency domain approach of FW-H formulation which is applicable to the computed domain is used as the governing equation for the noise prediction far downstream of the orifices.

The ANSYS-FLUENT software with parallel processing, version 16.1, is utilized to solve the filtered governing equations that are discretized by the finite volume method. The “SIMPLE” scheme is used for pressure-velocity coupling. For the momentum spatial discretization, “Bounded Central Differencing” scheme is applied and “Least Squares Cell Based” scheme, “PRESTO!” scheme, and “QUICK” scheme are used for gradient spatial discretization, pressure spatial discretization and
volume fraction, respectively. The “Bounded Second Order Implicit” formulation is deployed for the
temporal discretization[11].

When meshing the computational domain, the multi-block structured hexahedral mesh created by
ANSYS Meshing tools is used. The mesh is refined at the walls to capture the boundary layer. The
grid is concentrated near the walls and the measured $y^+$ is around 1. As the grid moves further away
from the walls, it is enlarged by an expansion ratio of 1.2:1. The initial instability of the shear layer
controls the aero-acoustic interaction, therefore the resolution close to the edge can be a crucial
parameter and the mesh is refined around the orifice[16]. In the radial direction, the grid size is refined
as well at the orifice wall and in the area of the jet downstream of the orifice. Table 2 shows the
features of the computational grids. This choice of the grid size guaranteed grid-independent simulations.

| Table 2. Features of the computational grids. |
|---------------------------------------------|
| Number of Cells | $2.3 \times 10^6$ |
| Smallest cell size (m) | $1.48 \times 10^{-4}$ |
| Largest cell size (m) | $2.8 \times 10^{-3}$ |

High grid resolution with enough small time steps is applied to resolve the relevant fluctuations. The
unsteady calculations are performed with a time-step size of $5 \times 10^{-5}$ s. This time-step is sufficient
to keep the Courant number less than one for the majority of computational cells and accurately
resolve pressure spectra within human hearing range. Unsteady simulation parameters are listed in
table 3. At each case, the transient solution is run for 1000 time steps before enabling the acoustics
model to obtain a statistically steady-state solution. After the unsteady flow field under consideration
has become fully developed, the FW-H acoustics model is enabled and acoustic source data is
exported for 0.05s flow time (1000 time steps).

| Table 3. Unsteady simulation parameters. |
|----------------------------------------|
| Parameter | Value |
| Total simulated flow time (s) | 0.1 |
| Simulated flow time before enabling the acoustics model (s) | 0.05 |
| Time step(s) | $5 \times 10^{-5}$ |
| Number of total time steps | 2,000 |
| Number of time steps before enabling the acoustics model | 1,000 |
| Max. Number of iterations per time step | 50 |
| Average hardware operating time per case (s) | 518,400 |

3.2. Boundary and Initial conditions
The mass flow inlet boundary condition with pressure is employed at the inlet of the pipe and a
pressure outlet boundary condition is applied at the domain outlet. To simulate different cavitation
cases, values of the inlet mass flow rate and outlet pressure at different cases was calculated based on
the values presented by Testud et al. resulting from their experiments (table 4)[3]. The converged
steady-state results are applied as initial conditions for unsteady simulation.

4. Results and discussion
Table 4 shows the flow conditions for the developed cavitation and the super cavitation in the single-hole orifice experiments conducted by Testud et al.[3].

**Table 4. Flow conditions in the single-hole orifice [3].**

|                  | Developed cavitation | Super cavitation |
|------------------|----------------------|------------------|
| $U(\text{ms}^{-1})^a$ | 1.91                 | 2.38             |
| St. deviation($\text{ms}^{-1}$) | 0.06                 | 0.04             |
| $P_1(10^5 \text{ Pa})^b$ | 6.3                  | 9.2              |
| St. deviation($10^5 \text{ Pa}$) | 0.3                  | 0.3              |
| $P_2(10^5 \text{ Pa})^b$ | 2.7                  | 2.7              |
| St. deviation($10^5 \text{ Pa}$) | 0                    | 0                |
| $\sigma^c$      | 0.74                 | 0.41             |

$^a$ The Reynolds number is based on the pipe diameter and the water viscosity that varies from $2 \times 10^5$ to $5 \times 10^5$.

$^b$ The static pressures $P_1$ and $P_2$ are measured 11D upstream and 40D downstream of the orifice, respectively.

$^c$ Cavitation Number.

Figure 2 shows the vapor volume fraction at different flow times downstream of the single-hole orifice for the developed cavitation condition ($\sigma=0.74$) and the super cavitation condition ($\sigma=0.10$). A comparison between figure 2A and figure 2B clearly shows that for the super cavitation condition there are more bubbles downstream of the orifice. At 0.1s flow time, for the super cavitation condition the vapor volume fraction is around 1 in the majority of areas downstream of the orifice while for the developed cavitation condition, the vapor volume fraction is near one just around the jet formed at the orifice. In the super cavitation condition, vapor pockets are formed in the jet region and expand downstream of the orifice while in the developed cavitation condition bubbles are observed around the jet formed at the orifice.
Figure 2. (a) Vapor volume fraction at different flow times downstream of the single-hole orifice for the developed cavitation condition ($\sigma=0.74$): a) 0.015s, b) 0.05s, c) 0.07s, and d) 0.1s[17].

Figure 2. (b) Vapor volume fraction at different flow times downstream of the single-hole orifice for the super cavitation condition ($\sigma=0.10$): a) 0.015s, b) 0.05s, c) 0.07s, and d) 0.1s[17].
Figure 3 shows the power spectral density of the sound pressure at 28D downstream of the single-hole orifice for two conditions, $\sigma=0.74$ and $\sigma=0.10$. From the results, it can be concluded that the spectral characteristics of the sound pressure fluctuations are influenced by the cavitation condition. The level of power spectral density increases in the super cavitation condition ($\sigma=0.10$).

For both cases, the spectra show that most peaks formed at a frequency range from 2000 Hz to 5000 Hz. This includes the range which human hearing is more sensitive to sounds. The human hearing range is generally mentioned to be 20 Hz to 20 kHz, but it is far more sensitive to sounds between 1 kHz and 4 kHz [18].

In figure 4, sound pressure levels at 28D downstream of the single-hole orifice are presented for both developed cavitation and super cavitation conditions ($\sigma=0.74$ & $\sigma=0.10$). The trend of the graphs for both conditions is almost the same. In low frequencies (below 800 Hz) there are few peaks and troughs and generally the sound pressure level is not increased. In both cases, the sound pressure level starts to increase around 800 Hz and reaches its maximum peak around 2000 Hz. More peaks and troughs can be observed at this range (1 kHz to 2 kHz). The average sound pressure level doesn’t change between 2 kHz and 4 kHz and starts to decrease after 4 kHz. The trend of the graph for the developed cavitation condition ($\sigma=0.74$) is very close to that of the acoustical power spectra generated from the experimental data for the same cavitation number in the frequency range of 100-1000 Hz [3].

![Figure 3](image-url)

Figure 3. (a) Power spectral density of the sound pressure at 28D downstream of the single-hole orifice for the developed cavitation condition ($\sigma=0.74$)[17].
Figure 3. (b) Power spectral density of the sound pressure at 28D downstream of the single-hole orifice for the super cavitation condition ($\sigma=0.10$)\cite{17}.

While the sound pressure level is between 70 dB and 110 dB for the developed cavitation condition, it can be observed from figure 4 that for the super cavitation condition, the sound pressure level is higher between 85 dB and 125 dB. For both conditions, the sound pressure level is loud and broad in the range of 2-4 kHz. It is notable that this is the range which human hearing is more sensitive to sounds.
Figure 4. Sound pressure levels at 28D downstream of the single-hole orifice for both the developed cavitation and the super cavitation conditions ($\sigma=0.74$ & $\sigma=0.10$)[17].

5. Conclusions
In this work, the acoustical effects of cavitating flow through a single-hole orifice were investigated. The LES approach was applied to calculate characteristics of unsteady flow. The developed and super cavitation conditions were studied, and for each condition the sound pressure signals far downstream of the orifices were computed by the FW-H formulation. Then the power spectral density of the sound pressure and the sound pressure level far downstream of the orifices were calculated.

The results confirm that for both developed and super cavitation conditions, the spectral characteristics of the sound pressure fluctuations are influenced by the cavitation condition, and that the super cavitation condition generates significantly more noise than the developed cavitation condition. In a single-hole orifice, the sound pressure level is between 70 dB and 110 dB for the developed cavitation condition, and higher between 85 dB and 125 dB for the super cavitation condition. For both conditions, the sound pressure level is loud and broad in the range of 2-4 kHz. The results of this work demonstrate that a combined LES and FW-H formulation is an appropriate approach for acoustical study of cavitating orifices although there are some limitations with the FW-H method. In this work, the Navier-Stokes equations are solved for incompressible flow and the effect of compressibility is ignored in terms of noise generated by the cavitating flow. Further research should be performed to investigate the role of compressibility in noise generation at a cavitating orifice.

References
[1] Zhang J M, Yang Q, Wang Y R, Xu W L and Chen J G 2011 Experimental investigation of cavitation in a sudden expansion pipe J. Hydrodynamics 23 348-52
[2] Nagaya Y and Murase M 2012 Detection of cavitation with directional microphones placed outside piping Nuclear Eng. and Design 249 140-5
[3] Testud P, Moussou P, Hirschberg A and Aurégan Y 2007 Noise generated by cavitating single-hole and multi-hole orifices in a water pipe J. Fluids and Structures 23 163-89
[4] Testud P, Aurégan Y, Moussou P and Hirschberg A 2009 The whistling potentiality of an orifice in a confined flow using an energetic criterion J. Sound and Vibration 325 769-80
[5] Kudźma Z and Stosiak M 2015 Studies of flow and cavitation in hydraulic lift valve Arch. of Civil and Mechanical Eng. 15 951-61
[6] Quan K M, Avvaru B and Pandit A B 2011 Measurement and interpretation of cavitation noise in a hybrid hydrodynamic cavitating device AICHE Journal 57 861-71
[7] Kugou N, Matsuda H, Izuchi H, Miyamoto H, Yamazaki A and Ogasawara M 1996 Proc. ASME Fluids Eng. Div. Summer Meeting(New York) vol 1 p 457-62
[8] Takahashi K, Matsuda H and Miyamoto H 2001 Proc. fourth Int. Symp. on Cavitation (Pasadena) session A9.006 p 1
[9] Moussou P, Cambier C, Lachene D, Longarini S, Paulhiac L and Villouvier V 2001 Proc. ASME Pressure Vessels and Piping Division Conf. (Atlanta) vol 420–2 (New Yourk: ASME Press) p 99–106
[10] Moussou P, Caillaud L, Villouvier V, Archer A, Boyer A, Rechu B and Benazet S 2003 Proc. ASME Pressure Vessels and Piping Division Conf. (Cleveland) vol 465 (New Yourk: ASME Press) p 161–8
[11] ANSYS Inc. 2017 ANSYS-FLUENT (Canonsburg: ANSYS Inc.)
[12] Meneveau C and Katz J 2000 Scale-invariance and turbulence models for large-eddy simulation Annual Review of Fluid Mechanics 32 1-32
[13] Chung T J 2002 Computational Fluid Dynamics (Huntsville: Cambridge University Press)
[14] Olivares P A V 2009 Acoustic wave propagation and modeling turbulent water flows with acoustics for district heating pipes (Uppsala: Uppsala University)
[15] Schnerr G H and Sauer J 2001 Physical and numerical modeling of unsteady cavitation dynamics Fourth Int. Conf. on Multiphase Flow (New Orleans)
[16] Hofmans G C J, Ranucci M, Ajello G, Auregan Y and Hirschberg A 2001 Aero-acoustic response of a slit-shaped diaphragm in a pipe at low Helmholtz number-part 2: unsteady results J. Sound and Vibration 244 57-77
[17] Bashirzadeh Tabrizi A and Wu B 2017 Acoustics study of cavitating orifices through computational fluid dynamics J. Fluids Eng. under review
[18] Smith S W 1997 The Scientist and Engineer's Guide to Digital Signal Processing ( San Diego: California Technical Puplishing)