Numerical investigation of unsteady cavitating flow around a hydrofoil using the PANS method

To cite this article: X Wang et al 2018 IOP Conf. Ser.: Earth Environ. Sci. 163 012076

View the article online for updates and enhancements.

Related content

- A new nonlinear turbulence model based on Partially-Averaged Navier-Stokes Equations
  J T Liu, Y L Wu, C Cai et al.

- Cavitation on hydrofoils with sinusoidal leading edge
  H Johani

- Unsteady Cavitating Flow around a Hydrofoil Simulated Using the Partially-Averaged Navier—Stokes Model
  Ji Bin, Luo Xian-Wu, Wu Yu-Lin et al.
Numerical investigation of unsteady cavitating flow around a hydrofoil using the PANS method

X Wang¹, Q Liu², Y J Tian¹ and P Xue¹

¹China Institute of Water Resources and Hydropower Research, Beijing 100038, China
²China Ship Development and Design Center, Wuhan 430072, China. E-mail: liuqi67@whu.edu.cn

Abstract. The unsteady cavitating flows around a NACA66 hydrofoil with chord length 150 mm and the attack angle of 6 degrees was numerically simulated using two different turbulence models: the standard k-ε turbulence model and the PANS model. The results show that the standard k-ε turbulence model over predicts the eddy viscosity in the rear part of the cavity and fails to predict the unsteady shedding dynamics. However, it is found that the PANS model can accurately capture the general features of unsteady cavitating flow and the predicted shedding frequency agrees fairly well with the corresponding experimental data. Based on the predicted numerical results, the vorticity transport equation was implied to further analyze the strong interaction between the cavitation and vortex. Thefoundings in this study can provide further insight into the mechanism of cavitation shedding dynamics.

1. Introduction

Cavitation is a two phase flow phenomenon which involves liquid-vapor phase change process and it usually occurs in liquid medium as its local pressure falls below the saturation vapor pressure[1-2]. It is well known that cavitation involves strong interactions between turbulence and unsteady shedding dynamics while the corresponding physical mechanisms are not yet fully understood. Recently, special efforts have been made to study the cavitation phenomenon due to its significant influence on many engineering applications in ship propellers, pump systems and turbine machines [3-5].

The numerical study of unsteady cavitating flow has become an effective method to investigate the cavitation shedding dynamics mechanism because it is able to provide detailed flow information which is very difficult or just cannot obtained experimentally. As for the numerical simulations of unsteady cavitating flows, the turbulence model plays a major role in the successful predictions of shedding dynamics. The widely applied Reynolds Averaged Navier-Stokes (RANS) equation approach has been used to consider turbulence effects in cavitating flows. However, it is found that the RANS models fail to capture the re-entrant jet induced by the adverse pressure gradient due to the over-prediction of turbulence viscosity in the cavity closure region[6]. Both the direct numerical simulation (DNS) and the large eddy simulation (LES) may provide an effective way to capture the unsteadies of cavitating flows, but their extremely high demands for large computing resources has generally restricted their practical applications. Therefore, some hybrid models have been proposed to simulate the cavitating turbulence flows, such as DCM model [7], FBM model[8] and FBDCM model [9]. Girimaji[10] proposed the PANS model as a bridging method from the RANS to DNS aiming to resolve large scale structures for non-cavitating flows at reasonable computational resources. Also, it has been reported
by many researchers that the PANS method shows much potential in simulating unsteady cavitating flows\cite{5,11,12}.

In the present paper, comparative study of RANS method and PANS method was conducted to further validate the merits of PANS model in simulating the shedding dynamics of cavitating flow. The periodic features of cavity development, shedding and collapse over a NACA66 hydrofoil were investigated. The vortex transport equation was used to obtain a deep insight in the interactions between the cavitation development and the vortex formation.

2. PANS method and cavitation model
The present numerical framework has based on the homogenous mixture assumption that the liquid-vapor multiphase is considered as a single fluid, and the slip velocity between the two phases is also ignored. Thus, the liquid and vapor phases in the cavity share the same pressure and velocity fields. Therefore, the governing equations for conservation of mass and momentum can be described as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right)\frac{\partial u_i}{\partial x_j} + \mu_t \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \mu_t \frac{\partial u_k}{\partial x_k} \delta_{ij}\right]
\]

where \(u_i\) and \(u_j\) represents the velocities in the \(x_i\) and \(x_j\) coordinate directions, respectively. \(\rho\) and \(P\) are the pressure and mixture density. \(\mu\) is the laminar viscosity of the homogenous mixture, and \(\mu_t\) is the turbulent viscosity which is predicted by turbulence model.

The fluid mixture properties are calculated as:

\[
\rho = \alpha_l \rho_l + \left(1 - \alpha_v\right) \rho_v
\]

\[
\mu = \alpha_l \mu_l + \left(1 - \alpha_v\right) \mu_v
\]

where subscripts \(l\) and \(v\) represent liquid and vapor, respectively. \(\alpha_v\) stands for the vapor volume fraction which is determined by the following transport-based equation:

\[
\frac{\partial \left(\rho \alpha_v\right)}{\partial t} + \frac{\partial \left(\rho \alpha_v u_j\right)}{\partial x_j} = \dot{m}^+ - \dot{m}^-
\]

where \(\dot{m}^+\) and \(\dot{m}^-\) are the mass transfer source term which is closed by the cavitation model.

2.1. Turbulence model
The standard \(k-\varepsilon\) turbulence model is one of the most widely applied turbulence models. The transport equations of turbulence kinetic energy \(k\) and the turbulence dissipation rate \(\varepsilon\) can be written as:

\[
\frac{\partial \left(\rho k\right)}{\partial t} + \frac{\partial \left(\rho k u_j\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right) \frac{\partial k}{\partial x_j}\right] + P_t - \rho \varepsilon
\]

\[
\frac{\partial \left(\rho \varepsilon\right)}{\partial t} + \frac{\partial \left(\rho \varepsilon u_j\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \mu_t\right) \frac{\partial \varepsilon}{\partial x_j}\right] + C_{c1} \rho \frac{\varepsilon}{k} - C_{c2} \rho \frac{\varepsilon^2}{k}
\]

where \(\mu_t\) is the turbulent viscosity calculated by \(\mu_t = C_{\mu} \rho k^2/\varepsilon\) with \(C_{\mu} = 0.09\), \(\sigma_k\) and \(\sigma_\varepsilon\) stand for the turbulent Prandtl number for \(k\) and \(\varepsilon\), and their values are 1.3 and 1.0, respectively. The other two empirical constants are \(C_{c1} = 1.44\) and \(C_{c2} = 1.92\).

The PANS model can bridge the physical resolution from RANS to DNS by specifying two control parameters \(f_k\) and \(f_\varepsilon\) which are defined as:

\[
f_k = \frac{k_u}{k}, \quad f_\varepsilon = \frac{\varepsilon_u}{\varepsilon}
\]
where $f_k$ is the ratio of resolved-to-total kinetic energy, and $f_\varepsilon$ is the ratio of resolved-to-total dissipation. The transport equations of unresolved kinetic energy $k_u$ and unresolved dissipation rate $\varepsilon_u$ in PANS method can be written in the classical RANS form:

$$
\frac{\partial (\rho k_u)}{\partial t} + \frac{\partial (\rho k_u u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{ku}} \right) \frac{\partial k_u}{\partial x_j} \right] + P_{ku} - \rho \varepsilon_u \tag{8}
$$

$$
\frac{\partial (\rho \varepsilon_u)}{\partial t} + \frac{\partial (\rho \varepsilon_u u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon u}} \right) \frac{\partial \varepsilon_u}{\partial x_j} \right] + C_{\varepsilon 1} \frac{P_{ku}}{k_u} \varepsilon_u - C_{\varepsilon 2} \rho \frac{\varepsilon_u^2}{k_u} \tag{9}
$$

where the eddy viscosity of PANS $\mu_{tu}$ is computed by $\mu_{tu} = C_\mu \rho k_u^2 / \varepsilon_u$, and the empirical constants in PANS model are defined as:

$$
\sigma_{ku} \equiv \sigma_k \frac{f_k^2}{f_\varepsilon} \tag{10}
$$

$$
\sigma_{\varepsilon u} \equiv \sigma_\varepsilon \frac{f_k^2}{f_\varepsilon} \tag{11}
$$

$$
C_{\varepsilon 2}^* \equiv C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon} \left( C_{\varepsilon 2} - C_{\varepsilon 1} \right) \tag{12}
$$

when $f_k$ and $f_\varepsilon$ are both set as unity, the PANS model will revert back to the standard k-\varepsilon model, the limiting case of $f_k = 0$ and $f_\varepsilon = 0$ is corresponding to DNS. For high Reynolds number flows, the value of $f_\varepsilon$ is recommended to set as unity [10]. It is noted that the different values of $f_k$ controls the resolution of the flow fields and a smaller $f_k$ is capable of resolving a larger number of scales of flow motions. It has been reported previously that the PANS model with $f_k = 0.2$ is suitable for simulating unsteady cavitating flows [13]. Therefore, the PANS model with $f_k = 0.2$ and $f_\varepsilon = 1.0$ was adopted in this work.

2.2. Cavitation model

The source terms $\dot{m}^+$ and $\dot{m}^-$ in equation (4) stand for the total effects of evaporation and condensation during the cavitation process and is described using the Rayleigh-Plesset equation[14]:

$$
R_b \frac{d^2 R_b}{dt^2} + \frac{3}{2} \left( \frac{dR_b}{dt} \right)^2 = \frac{p_b - p}{\rho_b} - \frac{4 \mu_t}{\rho_b R_b} \frac{dR_b}{dt} - \frac{2 \sigma}{\rho_b R_b} \tag{13}
$$

where $R_b$ represents the single bubble radius, $p$ is the surface pressure of the spherical bubble and $p_b$ is the local pressure, and $\sigma$ is the coefficient of surface tension.

The famous Rayleigh relation can be derived from the Rayleigh-Plesset equation simply by ignoring the second order derivative term of the bubble radius, the viscous effect term and the surface tension force term. Thus, the following equation can be obtained:

$$
\frac{dR_b}{dt} = \text{sign} \left( \frac{p_b}{\rho_b} \right) \left[ \frac{2}{3} \left( \frac{p_v(T)}{\rho_b} - p \right) \right]^{1/2} \tag{14}
$$

To close the equations, Sauer and Schnerr[15] introduced an equation to relate the vapor volume fraction $\alpha_v$ in equation (4) with the bubble radius $R_b$:

$$
\alpha_v = N_b \frac{4 \pi}{3} R_b^3 \left( 1 + N_b \frac{4 \pi}{3} R_b^3 \right)^{-1} \tag{15}
$$

Where $N_b$ is the bubble number density which means that there are $N_b$ bubbles per unit volume. Therefore, the detailed cavitation model can be written as
$$m^+ = \frac{\rho_s \rho_l}{\rho} \frac{3 \alpha_s}{R_0} \left( \frac{2 \max (p_s - p, 0)}{3 \rho_l} \right)^{1/2}$$  \hspace{1cm} (16)$$

$$m^- = \frac{\rho_s \rho_l}{\rho} \frac{3 \alpha_s}{R_0} \left( \frac{2 \max (p - p_s, 0)}{3 \rho_l} \right)^{1/2}$$  \hspace{1cm} (17)$$

where $p_s$ is the saturation vapor pressure.

It is noted that the bubble number density $N_b$ is the only empirical parameter in this cavitation model. In this study, $N_b$ is fixed at its default value $10^{13}$ as widely adopted by previous studies [16-17].

3. Simulation setup

Cavitating flows around a NACA66 hydrofoil with chord length $C = 150$ mm which was extensively studied experimentally by Leroux et al. [18] was analyzed numerically in the present study. The computational domain and boundary conditions were specified according to the experimental setup as shown in Figure 1. The inflow velocity was fixed at 5.33 m/s and the pressure outlet boundary condition was applied at the domain outlet. The outlet static pressure was determined to ensure the cavitation number ($\sigma = (p_{out} - p_v)/(0.5 \rho V^2)$) was 1.25. The total length of the computational domain was $10C$ with the inlet boundary located at 2.5 $C$ upstream of the hydrofoil centroid and the outlet boundary located at 6.5 $C$ downstream of the hydrofoil centroid.

![Figure 1. 2D computation domain and boundary conditions](image)

The above mentioned time-dependent governing equations were finally discretized in both space and time domains based upon the Finite Volume Method along with SIMPLE numerical algorithm. The second-order accurate central differential scheme was used to discretize the diffusion terms. The QUICK-type scheme was adopted to approximate the vapor volume fraction equation and the second-order upwind scheme was implemented to discretize the other convective terms. The bounded second-order implicit formulation was used to calculate the transient term. The time step was specified as $\Delta t = 10^{-4}$ s to resolve the real transient features of cavitating flows.

Three mesh resolutions were generated for mesh independence analysis with three different node numbers: 19529 nodes (Mesh 1), 33139 nodes (Mesh 2) and 78414 nodes (Mesh 3). For all the three meshes, the grids near the hydrofoil surface region was sufficiently refined to ensure that the values of $y^+$ lies between 30 and 90 to satisfy the near wall treatment requirement. Non-cavitating flow simulations were conducted to perform mesh independence analysis with standard $k$-$\varepsilon$ model. The corresponding pressure coefficient ($-C_p = (P_{ref} - P)/(0.5 \rho V^2)$) distributions around the hydrofoil were given in figure 2. It is demonstrated that the differences between Mesh 1 and Mesh 2 are obvious. However, the differences between Mesh 2 and Mesh 3 are negligible. Thus, Mesh 2 with 33139 nodes, as shown in figure 3, was selected as the final fluid mesh in the following simulations.
4. Results and discussion

The transient cavitating simulations in this study were started from unsteady non-cavitating flow fields to ensure numerical stability. The time history of the total vapor volume fraction $V_c$ calculated at each time step by the standard $k$-$\varepsilon$ turbulence model is shown in Figure 4 to better illustrate the unsteady features of cavitating flows, where $V_c$ is defined as:

$$V_c = \sum_{i=1}^{N} \alpha_i V_i$$  \hspace{1cm} (18)

where $N$ represents the total number of control volumes in the mesh domain, $\alpha_i$ stands for the vapor volume fraction in each single control volume with volume value of $V_i$. It is noted that the variations of vapor volume fraction is gradually damped out and the results turned out to be steady.

Figure 4. Calculated total vapor volume fraction by standard $k$-$\varepsilon$ model

Figure 5 shows the distributions of pressure, velocity, vapor volume fraction and eddy viscosity calculated by the standard $k$-$\varepsilon$ turbulence model. A stable cavity is attached to the hydrofoil without any obvious changes in volume as shown in Figure 5(c). However, the experimental results of Leroux et al. [18] shows that the cavitating flow is highly unstable, the attached sheet cavity gradually develops from the hydrofoil leading edge and subsequently is rolled up by the main flow as shown in Figure 6. Thus, the standard $k$-$\varepsilon$ model fails to capture the unsteady cavitating flow features of the studied case. It is noted that the eddy viscosity in the rear part region of the cavity is comparatively higher than the surroundings as shown in Figure 5(d) and such over-prediction of eddy viscosity suppresses the formation of re-entrant jet. Therefore, the sheet cavity cannot be entrained into the main stream.
Figure 5. Contours of (a) pressure, (b) velocity, (c) vapor volume fraction and (d) eddy viscosity predicted by standard $k$-$\varepsilon$ model

Figure 6. Experimental photographs of partial cavitation during one cycle [18]

Figure 7 shows the predicted time evolution of vapor volume fraction by the PANS method, and the shedding frequencies based up on the calculated vapor volume fraction is given in figure 8. As can be seen from the two figures, the process of cavitating flow is quasi-periodic and the most dominant frequency is about 3.51 Hz which agrees fairly well with the measured experimental value of 3.625 Hz by Leroux et al.[18].
Figure 7. Calculated total vapor volume fraction by PANS model

Figure 8. Spectral analysis of the total vapor volume fraction by PANS model

Figure 9 shows the time evolution of the instantaneous cavitation shedding processes by the PANS model in a typical periodic cycle at eight typical instants. It is found that the PANS model can capture the general unsteady features of cavitating flows and the results agree fairly well with the experimental observations as illustrated in Figure 6. Initially, a sheet cavity is attached to the leading edge and gradually growing up. And then, the cavity becomes thicker and its behavior gets unsteady. The attached cavity eventually becomes highly unstable and a cloud cavity subsequently is formed. The cloud cavity develops continuously and is convected into the downstream.

Figure 9. Sequences of predicted cavitation shedding in one typical cycle

Figure 10 illustrates the distributions of vorticity along the hydrofoil during one typical shedding cycle. It should be mentioned that the snapshots were taken at the same instants in Figure 9. It is noted that the vorticity distributions are quite related to the cavity developments. It is also demonstrated that there are strong interactions between the cavity and the vortex. The re-entrant jet originated near the
trailing edge can be identified as the regions of positive vorticity in the hydrofoil suction side surface. The re-entrant jet gradually develops in the upstream direction which leads to detachment of the sheet cavity. Therefore, it is confirmed that the PANS model can effectively reduce the eddy viscosity level in the rear part region of cavity and correctly capture the re-entrant jet behaviors.

The vorticity transport equation can be applied to better understand the mechanisms of the strong interactions between the cavitation and vortex. Usually, the vorticity transport equation of liquid-vapor two flows can be rearranged into the following form:

$$\frac{D\omega}{Dt} = (\omega \cdot \nabla)V - \omega(\nabla \cdot V) + \frac{\nabla \rho \times \nabla p}{\rho^2} + (v + v_t)\nabla^2 \omega$$  \hspace{1cm} (19)$$

where the only term on the left-hand side (LHS) stands for the total rate of the vorticity changes. The first term on the right-hand side (RHS) is called the vortex stretching term which represents the velocity gradient induced vorticity change. Obviously, this term equates zero for 2D cases. The second term on the RHS is the vortex dilatation term which accounts for the contributions of density changes to vorticity variations. And the third term on the RHS is called the baroclinic term which accounts for the effects of misaligned gradients of pressure and density to the rate of vorticity change. The last term on the RHS deals with the vorticity viscous diffusion and is ignored in the present study as the concerned flow is highly turbulent.

**Figure 10.** Contours of vorticity distributions in one typical cycle.

In this study, the corresponding vorticity equation can be rewritten as follows:

$$\frac{D\omega_z}{Dt} = \left[ -\omega(\nabla \cdot V) \right]_z + \left[ \frac{\nabla \rho \times \nabla p}{\rho^2} \right]_z$$ \hspace{1cm} (20)$$

$$\omega_z = \frac{\partial V_y}{\partial x} - \frac{\partial V_x}{\partial y}$$ \hspace{1cm} (21)$$
\[-[\omega (\nabla \cdot V)]_z = -\omega_z \left( \frac{\partial V_x}{\partial x} + \frac{\partial V_y}{\partial y} \right)\]  \hspace{1cm} (22)

\[\left[ \nabla \rho \times \nabla p \right]_z = \frac{1}{\rho} \left( \frac{\partial \rho}{\partial x} \frac{\partial \rho}{\partial y} - \frac{\partial \rho}{\partial y} \frac{\partial \rho}{\partial x} \right)\]  \hspace{1cm} (23)

**Figure 11.** The calculated dilatation term distributions in one typical cycle.

**Figure 12.** The calculated baroclinic term distributions in one typical cycle.
The predicted dilatation term distributions at different instants of time in one typical cycle are illustrated in Figure 11. By comparison with Figure 9 and Figure 11, it can be seen that the dilatation term equals to zero in the non-cavitation regions, while, in the cavitation regions, the dilatation term is either positive or negative. As can be seen in the vorticity transport equation, the dilatation term is directly related to the velocity divergence (\nabla \cdot V) which accounts for the impact of fluid compressibility. Thus, according to the dilatation term, the vorticity is enhanced when fluid is compressed in the vortex region and the vorticity is weakened when the fluid is expanded in the vortex region. Therefore, the dilatation term acts as the main contributor to the vorticity production in the quasi-periodic cavitation shedding process.

As shown in Figure 12, in the non-cavitation regions, the pressure and density gradients are identically aligned and thus the baroclinic term, that is proportional to \[ \nabla \rho \times \nabla p \] is also zero. In the cavity regions, the values of the baroclinic term are not identically equal to zero as the density and pressure gradients are not necessarily aligned. It should be noted that the non-zero values of the baroclinic term are mainly located at the cavity interface region and generally the intensity of the baroclinic term is much smaller than that of the dilatation term.

5. Conclusions
In this paper, the quasi-periodic unsteady cavitating flow over a NACA66 hydrofoil was simulated based on the standard k-\epsilon turbulence model and the PANS model. The main conclusions can be drawn as follows:

- The standard k-\epsilon turbulence model over-predicts the eddy viscosity in the rear part of the cavity and fails to capture the unsteady features of the cavitating flow. The PANS model can successfully predict the quasi-periodic features of cavity development, shedding and collapse.
- The cavitation shedding frequency predicted by PANS model agrees reasonably well with the measured results. The re-entrant jet can be captured by the PANS model as well.
- The PANS model can well describe the strong interactions between the cavitation and vortex. Further analysis reveals that the dilatation term is the main contributor to the vorticity production in the cavitation shedding process and the baroclinic term is mainly located at the cavity interface region.

Acknowledgments
This work is supported by the IWHR Research and Development Support Program (HM0145B212015, HM0145B212016, HTJD1003B012017 and HM0145B412017).

Reference
[1] Arndt, R.E.A. 1981 Annu. Rev. Fluid Mech. 13 273
[2] Long X, Liu Q, Ji B and Lu Y 2017 Int. J. Heat and Mass Transfer 109 879-93
[3] Ji B, Luo X, Wu Y, Peng X and Duan Y 2013 Int. J. Multiph. Flow 51 33-43
[4] Long X, Cheng H, Ji B and Arndt R E A 2017 Ocean Eng. 137 247-61
[5] Ji B, Luo X, Wu Y, Peng X and Xu H 2012 Int. J. Heat and Mass Transfer 55 6582-88
[6] Ji B, Luo X, Arndt R E A and Wu Y 2014 Ocean Eng. 87 64-77
[7] Coutier-Delgosha O, Fortes-Patella R and Reboud J L 2002 J. Turbul 3 58
[8] Wu J, Wang G and Shyy W 2010 Int. J. Numer. Methods Fluids 49 739-61
[9] Huang B, Wang G and Zhao Y 2014 J. Hydrol, Ser B 26 26-36
[10] Girimaji S S 2006 J. Appl. Mech. 73 413
[11] Hu C, Wang G, Chen G and Huang B 2014 Sci. China. Phys. Mech. 57 1967-76
[12] Pan D Z, Zhang D S, Wang H Y, Shi W D and Shi L 2015 IOP Conf. Ser. Mater. Sci. Eng 72 22005
[13] Huang B and Wang G 2011 J. Hydrol, Ser B 23 26-33
[14] Brennen C E 1995 Cavitation and bubble dynamics (Oxford University Press) pp 609-17
[15] Sauer J and Schnerr G H 2000 Proc. FEDSM'00 (Boston: USA) pp 1-7
[16] Sauer J and Schnerr G H 2001 *J. Appl. Math. Mech* **81** 561-62
[17] Li Z R, Pourquie M and Terwisga T J 2010 *J. Hydrol, Ser B* 770-77
[18] Leroux J B, Astolfi J A, and Billard J Y 2004 *J. Fluids Eng* **126** 94-101