Evaluation of turbulence models for the numerical prediction of transient cavitation around a hydrofoil

To cite this article: W D Shi et al 2013 IOP Conf. Ser.: Mater. Sci. Eng. 52 062013

View the article online for updates and enhancements.

Related content
- Filter-based density correction model for turbulent cavitating flows
  B Huang, G H Chen, J Zhao et al.
- Evaluation of a Filter-Based Model for Computations of Cavitating Flows
  Huang Biao and Wang Guo-Yu
- Evaluation of turbulence models on roughened turbine blades
Evaluation of turbulence models for the numerical prediction of transient cavitation around a hydrofoil

W D Shi, G J Zhang and D S Zhang

Research Center of Fluid Machinery Engineering and Technology, Zhenjiang, 212000, China,

E-mail: 846763679@qq.com

Abstract. The objective of this paper is to evaluate the predictive capability of three turbulence models for the simulation of unsteady cavitating flows around a 2D Clark-y hydrofoil. Three turbulence models were standard k-\(\varepsilon\) model, hybrid model of density correction model (DCM) and filter-based model (FBM) and an improved partially-averaged Navier-Stokes model (PANS) based on k-\(\varepsilon\) model. Using the above-mentioned turbulence models and a homogeneous cavitation model, the unsteady cloud cavitating flows around the hydrofoil were numerically simulated and the time evolutions of cavity shape and lift evolutions over time were obtained. The results with comparison to a tunnel experiment data show that the hybrid model and PANS model can accurately capture unsteady cavity shedding details, fluctuation frequency and amplitude of lift and drag. The k-\(\varepsilon\) model has a poor agreement with the real experimental visualizations and this is mainly attributed to an over prediction of the turbulent viscosity in the rear part of the cavity, which limits the reentrant jet fully reaching the leading edge. The adverse pressure gradient plays an important role in the progression of the reentrant jet. Both the shock wave generated by the collapse of the cloud cavity and the growth of attached sheet cavity contribute to the increase of adverse pressure gradient.

1. Introduction
Cavitating flow is a kind of common phenomenon occurring in hydraulic machineries. The instability of cavitation often leads to some serious problems, such as noise, fluctuation, vibration and erosion. So it is necessary to study the unsteady cavitating flow. Noticeable efforts have been made to investigate cavitating flows numerically and experimentally in recent years [1-3]. The results show that in computations, the turbulence models play very important roles in the simulations of the cavitating flow. Coutier D. [4] reduced the turbulent viscosity by taking into account the compressibility of vapor-water mixture and obtained a satisfactory result compared to the experiment in Venturi. An actively pursued route to simulate unsteady cavitating flows is the large eddy simulations (LES) approach and Bensow [5] have used LES simulate the cloud cavitation successfully. However, it is difficult to find a grid independent LES solution [6]. An attractive aspect is that boundary layers can be calculated by RANS models and the Smagorinsky model is used for transient calculation of the large-scale turbulent structures away from the solid wall. According to this idea, Johansen [7] have formulated a filter-based model (FBM) blending RANS and LES. Subsequently, Wu [8] applied it to study could cavitation around a 2D Clark-y hydrofoil successfully. Girimaji [9] proposed a PANS model realizing seamless transition from RANS to DNS and predicted a reasonable result.
In this work, we evaluated the predictive capability of three turbulence models for the simulation of unsteady cavitating flows and revealed the cause of reentrant jet which leads to the instability of cloud cavity.

2. Numerical Method

2.1. Governing Equations

In CFX the homogeneous mixture flow is used for our problem. The vapor/liquid two-phase mixture model assumes the fluid to be homogeneous, so the multiphase fluid components share the same velocity and pressure. The continuity, momentum and mass transfer equations for the mixture flow are

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

(1)

\[
\frac{\partial (\rho_m u_j)}{\partial t} + \frac{\partial (\rho_m u_j u_j)}{\partial x_j} = \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\left(\mu + \mu_l\right) \frac{\partial u_j}{\partial x_j} + \frac{2}{3} \frac{\partial u_j}{\partial x_j}\right)
\]  

(2)

\[
\frac{\partial \rho \alpha_i}{\partial t} + \frac{\partial \rho \alpha_i u_j}{\partial x_j} = m
\]  

(3)

In the above equations, \(\rho_m = \rho_l (1 - \alpha_v) + \rho_v \alpha_v\), \(\mu = \mu_l (1 - \alpha_v) + \mu_v \alpha_v\), \(\rho_m\) represents the mixture density, \(\rho_l\) and \(\rho_v\) the liquid density and vapor density, \(\alpha_v\) the water volume fraction, \(u\) the time averaged mixture velocity, \(p\) the time averaged pressure, \(\mu\) the eddy viscosity, \(m\) the inter-phase mass transfer rate due to cavitation which can be modelled using a cavitation model.

2.2. Cavitation Model and Turbulent Correction

Cavitation model is a mathematical model describing conversion between the liquid and vapor phases. The model used in the paper is based on the simplified Rayleigh-Plesset equation for bubble dynamics, and the source term is as follows,

\[
m = \begin{cases} 
-F_v \frac{2 \rho_m (1 - \alpha_v) \rho_s}{R_B} \left( \frac{2}{3} \frac{p_s - p}{\rho_i} \right) & \text{if } p < p_s \\
F_e \frac{2 \rho_s \rho_v}{3} & \text{if } p > p_s 
\end{cases}
\]  

(4)

In the above equations, \(r_{nuc}\) is the nucleation site volume fraction, \(p_v\) is the vaporized pressure, \(R_B\) is the bubble radius, \(F_v\) and \(F_e\) are two empirical coefficients for the evaporation and condensation processes, respectively. Further, we considered the effect of turbulent pressure fluctuation \(p_{turb}\) on vaporized pressure and modified \(p_v\) as follows,

\[
p_v = p_{sat} + \frac{p_{turb}}{2}, \quad p_{sat} = 0.39 \rho_n k
\]  

(5)

where \(p_{sat}\) is the saturated liquid vapor pressure, \(k\) is the turbulent kinetic energy.

2.3. Turbulence Models

In this work, we evaluate the predictive capability of three turbulence models for the simulation of unsteady cavitating flows. The standard k-\(\varepsilon\) model is native in CFX, while the other two models are added to CFX via user-defined functions. In the following we provide a brief description of the three different models considered in this study.

2.3.1. The k-\(\varepsilon\) model

The k-\(\varepsilon\) model is as follows:

\[
\frac{\partial (\rho u_k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\mu + \mu_l\right) \frac{\partial k}{\partial x_j} + G_k - \rho \varepsilon\right)
\]  

(6)

\[
\frac{\partial (\rho \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\left(\mu + \mu_l\right) \frac{\partial \varepsilon}{\partial x_j} + \frac{\varepsilon}{k} (C_1 G_k - C_2 \rho \varepsilon)\right)
\]  

(7)
\[ \mu_s = \rho C_\mu k^2 \epsilon \]  

where the model constants are: \( C_1 = 1.44, \ C_2 = 1.92, \ \sigma_\epsilon = 1.0, \ \sigma_\mu = 1.3, \ C_\mu = 0.085. \)

2.3.2. The hybrid model. The RNG k-\( \epsilon \) model was modified simply by taking into account the local compressibility of vapor-water mixture, mainly in the hydrofoil leading edge areas:

\[ \mu_{DCM} = \rho C_\mu k^2 \epsilon f_{DCM}, \quad \mu_{FBM} = \frac{\rho_s + \alpha_\epsilon (\rho - \rho_s)}{\rho_s + \alpha_\mu (\rho - \rho_s)} \]

A filtered Navier-Stokes model, originated from the RNG k-\( \epsilon \) turbulence model, is introduced, mainly in the hydrofoil trailing edge areas:

\[ \mu_{FBM} = \rho C_\mu k^2 \epsilon f_{FBM}, \quad f_{FBM} = \text{Min}(1, \frac{\Delta \epsilon}{k^{2/3}}), \quad \Delta = 1.05 \sqrt{\Delta y} [10] \]

where \( \Delta \) and \( \Delta \epsilon \) are the grid element lengths in x and y direction. In the region where the turbulence length scale is less than the filter size \( \Delta \), the RNG k-\( \epsilon \) model with a standard wall-function is used, otherwise the LES works. The filter-based model allows much coarser grids in the boundary layer compared to common LES methods.

Based on the above two improving method, a blending function \( \chi(\rho/\rho_s) \) is used to combine the above two turbulent viscosities in different cavitation areas, which is given as

\[ \mu_{\text{hybrid}} = \rho C_\mu k^2 \epsilon f_{\text{hybrid}}, \quad f_{\text{hybrid}} = \chi(\rho/\rho_s)f_{FBM} + [1 - \chi(\rho/\rho_s)]f_{DCM} \]

The hybrid function is shown as Figure 1.

2.3.3. The PANS model. The model provides a seamless transition from RANS to the direct numerical solution (DNS) as the unresolved-to-total ratios of kinetic energy \( (f_k) \) and its dissipation \( (f_\epsilon) \) are varied.

\[ f_k = k_u / k, \quad f_\epsilon = \epsilon_u / \epsilon \]

The turbulent governing equations in the PANS are identical to the RANS equations, but with different model coefficients, which are defined as

\[ \sigma_{u\epsilon} = \sigma_\epsilon f_k / f_\epsilon, \quad \sigma_{u\mu} = \sigma_\mu f_k / f_\epsilon, \quad C_\epsilon^* = C_\epsilon + f_k / f_\epsilon (C_\epsilon - C_\mu) \]

According to the previous works [11], \( f_\epsilon \) can be set as 1.0 in high Reynolds number flows and \( f_k = 0.4 \) taking into account the mesh resolution in the present study.

2.4. Grid and boundary conditions

A C-type orthogonal mesh which is fit for the foil rounded leading edge is used as shown in Figure 2. The element sizes were carefully selected to ensure that the non-dimensional \( y^+ \) value can satisfy the requirement of wall function. The final mesh had a total of 54326 hexahedral cells with \( y^+ \) between 15 and 60.

The hydrofoil chord length was \( C = 0.07m \); the attack angle was \( \alpha = 8^\circ \); the cavitation number was \( \sigma = 0.8 \); the inflow velocity was \( U_{in} = 10m/s \) and the corresponding Reynolds number was \( 7\times10^5 \); the time step was set as \( \Delta t = 0.1ms \). The boundary conditions were as follows: a uniform velocity at the domain inlet, static pressure at the outlet. A no-slip boundary condition is used at both the upper and lower walls, also at the hydrofoil surface.

---

**Figure 1.** The hybrid function distribution.  
**Figure 2.** Computational grids near the hydrofoil.
3. Results and Discussions

The cavity shapes of five typical instants during one cavitating cycle obtained by the experimental observations [12] and numerical simulations are shown in Figure 3. The black and white correspond to vapor and water. According to experimental observations, in this condition the flow is characterized by unsteady cavitation behavior. However, the k-ε model simulates a quasi-steady behavior of the sheet cavitation. This may be mainly attributed to an over-prediction of the turbulent viscosity in the rear part of the cavity. The reentrant jet is the main trigger for unstable cloud cavitation, while the over-prediction of turbulent viscosity makes it not to have sufficient momentum to reach the leading edge, which prevents the breaking and shedding of the sheet cavity.

![Figure 3](image)

Figure 3. Time evolution of cloud cavitation for the experimental observations and numerical results.

As shown in Figure 3(c) and (d), the unstable cloud cavitation behavior is simulated successfully by the hybrid model and PANS model. Based on the above predicted results, we can see that the evolution of the cavitation flow in each cycle can be divided into the following steps: (a) The attached sheet cavity grows slowly up to the maximum lengths about 80% of the chord; (b) The reentrant jet flowing upstream cuts the cavity into two parts: attached sheet cavity in the foreside of the cavity and shedding vortex structure in the rear region; (c) The attached sheet cavity shrinks sequentially towards the leading edge and the shedding vapor cloud is rolled up and convects downstream following the main flow; (d) The shedding vapor clusters collapse in high pressure region while the residual cavity grows again.

Compared to the standard k-ε model, the hybrid model and PANS model correctly capture the behavior of reentrant jet. The hybrid model not only takes into account the local compressibility of vapor-water mixture mainly in the hydrofoil leading edge areas, but also simulates more accurately the large-scale vortex shedding using LES method by introducing a filter size in the rear region. The PANS model modifies the governing equations by changing $f_k$ value, realizing the smooth transition from RANS to DNS. As the $f_k$ decreases, the predicted cavitating flow becomes unsteady due to resolving more scale of the turbulent kinetic energy. So the results agree well with the experimental observations.

Figure 4 presents the evolutions of the lift coefficient given by calculations and experiment [13] during 3 cavitation cycle. The average lift and drag coefficients and oscillation cycle compared with experimental data are also given in Table 1. The hybrid model and PANS model both give a good agreement with experimental results concerning the oscillation frequency and the mean fluctuating values, while the k-ε model has a larger difference. The lift force shows a violent oscillatory behavior.

![Figure 4](image)

Table 1. The average lift and drag coefficients and oscillation cycle

<table>
<thead>
<tr>
<th></th>
<th>Lift coefficient</th>
<th>Drag coefficient</th>
<th>Oscillation cycle/ ms</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-ε model</td>
<td>0.682</td>
<td>0.109</td>
<td>48.4</td>
</tr>
<tr>
<td>hybrid model</td>
<td>0.743</td>
<td>0.126</td>
<td>38.8</td>
</tr>
<tr>
<td>PANS model</td>
<td>0.718</td>
<td>0.121</td>
<td>36.4</td>
</tr>
<tr>
<td>Exp.</td>
<td>0.76</td>
<td>0.119</td>
<td>41</td>
</tr>
</tbody>
</table>
The lift coefficient shows an increase during cavity growth up to a maximum, then a high-frequency fluctuation due to small-scale cavity detachments. When the main cloud detaches from the suction side, the lift decreases abruptly.

Some discrepancy is also noticed. The lift coefficients in the experiment seem to be gentler than data predicted by numerical simulations. The sampling frequency of pressure transducer may be a significant factor. The unstable cloud cavitation can induce a very high-frequency pressure fluctuation and the pressure transducer may omit some peak values resulting in a gentler lift curve.

The reentrant jet should be responsible to the instability of cloud cavitation and the turbulent viscosity has a great influence on reentrant jet behavior. Figure 5 shows the time-averaged turbulent viscosity distributions predicted by different models. The k-ε model gives an incredible turbulent viscosity region in the rear part of the cavity which hinders flowing towards the leading edge of the reentrant jet notably. As shown in Figure 5(b) and (c), the hybrid model and PANS model strongly reduce the turbulent viscosity mainly in the rear part of the cavity and predict reasonable time evolutions of cavity shape compared to experimental observations.

We used the simulating results of the hybrid model to reveal the cause of reentrant jet in consideration of its nice predictive capability. Figure 6 shows the pressure distributions at five representative instants. The t moment corresponds to about 43% cycle in Figure 3(c). As shown in Figure 6, the adverse pressure gradient in the cavity wake increases when cavity grows. The shedding cloud cavities collapse downstream at t+2.3ms which results in high pressure regions and the pressure grows to the maximum at t+2.5ms leading to a pressure wave that propagates in all directions and particularly upstream. When the adverse pressure gradient is strong enough to overcome the weaker momentum of the flow, the reentrant flow forms. The enhanced pressure gradient pushes the reentrant flow upstream and the reentrant flow encounters with the main flow at some position which causes a vortex in the rear part of the cavity and then the cavities detach from the suction side due to the effect of vortex. Based on the above analyses, we can know that the adverse pressure gradient plays an important role in the progression of the reentrant jet. Both the shock wave generated by the collapse of the cloud cavity and the growth of attached sheet cavity contribute to the increase of adverse pressure gradient.

**Figure 4.** Variation of lift coefficient with time

**Figure 5.** Time averaged eddy viscosity contours with time due to the cavity growth, cloud shedding and vapor collapse. The lift coefficient shows an increase during cavity growth up to a maximum, then a high-frequency fluctuation due to small-scale cavity detachments. When the main cloud detaches from the suction side, the lift decreases abruptly.

Some discrepancy is also noticed. The lift coefficients in the experiment seem to be gentler than data predicted by numerical simulations. The sampling frequency of pressure transducer may be a significant factor. The unstable cloud cavitation can induce a very high-frequency pressure fluctuation and the pressure transducer may omit some peak values resulting in a gentler lift curve.

The reentrant jet should be responsible to the instability of cloud cavitation and the turbulent viscosity has a great influence on reentrant jet behavior. Figure 5 shows the time-averaged turbulent viscosity distributions predicted by different models. The k-ε model gives an incredible turbulent viscosity region in the rear part of the cavity which hinders flowing towards the leading edge of the reentrant jet notably. As shown in Figure 5(b) and (c), the hybrid model and PANS model strongly reduce the turbulent viscosity mainly in the rear part of the cavity and predict reasonable time evolutions of cavity shape compared to experimental observations.

We used the simulating results of the hybrid model to reveal the cause of reentrant jet in consideration of its nice predictive capability. Figure 6 shows the pressure distributions at five representative instants. The t moment corresponds to about 43% cycle in Figure 3(c). As shown in Figure 6, the adverse pressure gradient in the cavity wake increases when cavity grows. The shedding cloud cavities collapse downstream at t+2.3ms which results in high pressure regions and the pressure grows to the maximum at t+2.5ms leading to a pressure wave that propagates in all directions and particularly upstream. When the adverse pressure gradient is strong enough to overcome the weaker momentum of the flow, the reentrant flow forms. The enhanced pressure gradient pushes the reentrant flow upstream and the reentrant flow encounters with the main flow at some position which causes a vortex in the rear part of the cavity and then the cavities detach from the suction side due to the effect of vortex. Based on the above analyses, we can know that the adverse pressure gradient plays an important role in the progression of the reentrant jet. Both the shock wave generated by the collapse of the cloud cavity and the growth of attached sheet cavity contribute to the increase of adverse pressure gradient.

**Figure 6.** The pressure distributions at five representative instants.
4. Conclusions
Three turbulence models have been introduced to simulate the unsteady cavitating flow around a 2D Clark-y hydrofoil. The results show that the quasi-periodic cavitation behavior is strongly influenced by the applied turbulence model. The results obtained with the standard k-ε model have a poor agreement with the real experimental visualizations and this is mainly attributed to an over prediction of the turbulent viscosity in the rear part of the cavity, which limits the reentrant jet fully reaching the leading edge. On the other hand, both the hybrid model and PANS model reproduce the vapor cloud shedding behavior observed in the experiment successfully. Compared to the standard k-ε model, the hybrid model and PANS model strongly reduce the turbulent viscosity which leads to a reasonable re-entrant jet behavior. According to the results predicted by the hybrid model, we could find that the adverse pressure gradient plays an important role in the progression of the re-entrant jet. Both the shock wave due to the collapse of cloud cavity and the effect of cavity growth enhance the adverse pressure gradient.

References