Investigation of cavitation around 3D hemispherical head-form body and conical cavitators using different turbulence and cavitation models

Mohammad-Reza Pendar, Ehsan Roohi *

High Performance Computing (HPC) Laboratory, Department of Mechanical Engineering, Faculty of Engineering, Ferdowsi University of Mashhad, P.O. Box 91775-1111, Mashhad, Iran

A R T I C L E   I N F O

Article history:
Received 18 January 2015
Accepted 7 December 2015

Keywords:
Conical cavitator
Hemispherical head-form body
Supercavitation
Volume-of-Fluid (VOF) method
OpenFOAM

A B S T R A C T

In this paper, cavitation and supercavitation around 3D hemispherical head-form body and a conical cavitator were simulated. Dynamic and unsteady behaviors of cavitation were solved using large eddy simulation (LES) and $k-\omega$ SST turbulence models, as well as Kunz and Sauer mass transfer models. In addition, the compressive volume of fluid (VOF) method is used to track the cavity interface. Simulation is performed under the framework of the OpenFOAM package. The main contribution of this work is to present a correlation between the cavity length and diameter for hemispherical head-form bodies for the first time. Moreover, we provide a detailed comparison between different turbulence and mass transfer models over a broad range of cavitation numbers, especially in small cavitation numbers, including $\sigma=0.07$, 0.05, 0.02 for two cases, which is not reported previously. Our numerical results are compared with the available experimental data and a broad set of analytic relations for the cavity characteristics such as cavity length and diameter with suitable accuracy. Discussions on boundary layer separation and re-entrant jet behavior, which play a significant role in the bubble shedding in the cavity closure region, are presented.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Cavitation is a multi-phase and complex physical phenomenon defined as the formation of vapor bubbles within a liquid when the pressure locally drops below the saturated vapor pressure (Brennen, 1995). As a useful phenomenon, cavitation attracted the attention of many researchers in the past decades. Cavitation usually appears over underwater vehicles such as underwater vehicles, submarine, hydrofoils and marine propeller blades. Formation of cavity cloud significantly increases the performance of the marines by reducing the viscous drag. Cavitation is a three-dimensional and periodic phenomenon that exhibits unsteady dynamic behaviors such as periodic shedding of the cavity cloud and growth and rapid collapse of bubbles (Wang and Ostoja, 2007). Cavitation could be categorized by a dimensionless number, i.e., $\sigma = (P_\infty - P_f)/0.5pU_c^2$ that is called cavitation number. When one decreases the cavitation number, i.e., via increasing the velocity of the moving body further, supercavitation will occur which consists of a long and steady cavity region.

Precise simulation of cavitation phenomenon requires an accurate mass transfer model, a surface reconstruction scheme for capturing the sharp cavity interface as well a suitable turbulence model. Different kinds of mass transfer model can be used for cavitation modeling. Famous cavitation models based on semi-analytical approaches were derived by Merkle et al. (1998), Kunz et al. (2000), Sauer (2000), Yuan et al. (2001) and Singhal et al. (2002).

Among different surface reconstruction schemes, Volume of Fluids (VOF) method was extensively used to describe the phase transition mechanism between liquid and vapor phases that both exist in cavitating flows. For instance, Passandideh Fard and Roohi (2008), Shang, (2013), Roohi et al. (2013), Yu et al. (2014) and Kim and Lee (2015) used VOF method to simulate cavitation for different sets of geometries. This method predicts the cavity interface accurately.

Selection of an appropriate turbulence model is another crucial issue for accurate simulation of the cavitation because the cavitation is an unsteady phenomenon usually occurring in high Reynolds number flows. Two turbulence approaches, large eddy simulation (LES) and $k-\omega$ SST have been most widely used to simulate cavitating flow. LES regularly allows for medium-scale to small-scale energy transfer that can capture flow mechanisms with much detail for accurate prediction of the cavitation.

Literature survey shows that numerical simulation of cavitation attracted the attention of researchers during the last decades. Kunz et al. (2000) considered cavitation around...
Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>Unresolved transport term in LES</td>
</tr>
<tr>
<td>C_x,C_y</td>
<td>LES empirical constant coefficients</td>
</tr>
<tr>
<td>C_x,C_y</td>
<td>Schröder–Sauer mass transfer model constants</td>
</tr>
<tr>
<td>CD_u0, Cprod</td>
<td>Positive portion of the cross-diffusion</td>
</tr>
<tr>
<td>C_d0, C_k</td>
<td>Kunz mass transfer model constants</td>
</tr>
<tr>
<td>C_d0</td>
<td>Constant in the drag coefficient for a conical cavitator</td>
</tr>
<tr>
<td>C_x, C_y</td>
<td>Functions (given by Eqs. (36) and (37))</td>
</tr>
<tr>
<td>D</td>
<td>Cavity diameter</td>
</tr>
<tr>
<td>d</td>
<td>Cavitator diameter</td>
</tr>
<tr>
<td>D_D</td>
<td>Rate of strain tensor</td>
</tr>
<tr>
<td>D_D</td>
<td>Eddy diffusivity tensor</td>
</tr>
<tr>
<td>F_1, F_2</td>
<td>Turbulence Functions (given by Eqs. (14) and (18))</td>
</tr>
<tr>
<td>G</td>
<td>Filter function</td>
</tr>
<tr>
<td>H</td>
<td>Height of the cone cavitator</td>
</tr>
<tr>
<td>I</td>
<td>Unit tensor</td>
</tr>
<tr>
<td>k</td>
<td>Kinetic energy</td>
</tr>
<tr>
<td>L</td>
<td>Cavity length</td>
</tr>
<tr>
<td>l</td>
<td>Liquid</td>
</tr>
<tr>
<td>m</td>
<td>Mass transfer rate between the phases</td>
</tr>
<tr>
<td>n_0</td>
<td>Initial number of bubbles</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
</tr>
<tr>
<td>R</td>
<td>Radius of Cylinder</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>R_b</td>
<td>Radius of bubbles</td>
</tr>
<tr>
<td>s</td>
<td>Viscous stress tensor</td>
</tr>
<tr>
<td>t_w</td>
<td>Mean flow time</td>
</tr>
<tr>
<td>U</td>
<td>Velocity magnitude</td>
</tr>
<tr>
<td>μSGS</td>
<td>Subgrid scale viscosity</td>
</tr>
<tr>
<td>X_i</td>
<td>Components of the Cartesian coordinate</td>
</tr>
<tr>
<td>y</td>
<td>Distance between the surface</td>
</tr>
<tr>
<td>σ</td>
<td>Cavitation number</td>
</tr>
<tr>
<td>∞</td>
<td>Free stream value</td>
</tr>
<tr>
<td>α_π</td>
<td>Cone half-angle</td>
</tr>
<tr>
<td>φ</td>
<td>Volumetric flux</td>
</tr>
<tr>
<td>ρ</td>
<td>Density</td>
</tr>
<tr>
<td>δ</td>
<td>Vapor</td>
</tr>
<tr>
<td>δ</td>
<td>Re-entrant jet length</td>
</tr>
<tr>
<td>v</td>
<td>Velocity vector</td>
</tr>
<tr>
<td>μ</td>
<td>Viscosity</td>
</tr>
<tr>
<td>Δ</td>
<td>Filter width</td>
</tr>
<tr>
<td>γ</td>
<td>Volume fraction</td>
</tr>
<tr>
<td>ω</td>
<td>Vorticity</td>
</tr>
<tr>
<td>μ_κ</td>
<td>Viscosity of the vortex</td>
</tr>
<tr>
<td>β, σ_m, δ, β, c_μ</td>
<td>Constant coefficients for the k-ω SST turbulence model</td>
</tr>
<tr>
<td>–</td>
<td>Averaging</td>
</tr>
</tbody>
</table>

submerged objects for the axisymmetric and steady state condition with multiphase computational fluid dynamics. They conveyed different parameter such as pressure distribution, drag coefficient and cavity shape and compared with the experimental data. Baradaran Fard and Nikseresht, (2012) simulated unsteady 3-D cavitating flows around a cone and disk cavitator. RANS equations and an additional transport equation for the liquid volume fraction are solved using a finite volume approach through the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm. For the implementation of the turbulent flow, k-ω SST model was used. The results are in good agreement with experimental data and analytical relations. Guo et al. (2011) simulated the cavitating flow around an underwater projectile with natural and ventilated cavitation based on the homogeneous equilibrium flow model, a mixture model for transport equation and a local linear low-Reynolds-number k-ε turbulence model. Shang (2013) simulated cavitation around the cylindrical submarine. They used K-ω SST for turbulence model, VOF method for the cavity interface reconstruction and the Sauer model for mass transfer to capture the cavitation mechanisms within broad ranges of cavitation numbers from 0.2 to 1.0. Park and Hyung (2012) simulated high-speed supercavitating flows around a 2-D symmetric wedge-shaped cavitator and hemispherical head form body using an unsteady Reynolds-averaged Navier–Stokes equations solver based on a cell-centered finite volume method. The computed result compared with an analytical solution and numerical results using a potential flow solver. Yu et al. (2014) simulated dynamic behaviors of cavitation over a 3-D projectile at the cavitation number σ = 0.58 based on LES, k-μ transport equation and VOF method with the Kunz model for the mass transfer. Evolution of cavitation in simulation was consistent with the experimental. Chen et al. (2015) investigated the collapse regimes of the cavitation on the submerged vehicles navigating with continuous deceleration in the range of 0.2 ≤ σ ≤ 0.5. A homogeneous equilibrium cavitation model that combined with the pressure–velocity–density coupling algorithm was used to simulate the cavitating flows. There are some recent works which report cavitation phenomena using advanced turbulence models. For example, Decaix and Goncalves (2013) used a compressible, multiphase, one-fluid RANS solver to study turbulent cavitating flows. Ji et al. (2013) simulated cavitating turbulent flow around hydrofoils by using the Partially-Averaged Navier–Stokes (PANS) method and a suitable mass transfer cavitation model. Their predicted cavity characteristic compared well with experimental data. Zhang and Khoo (2014) developed a pressure-based compressible-medium numerical method to perform computations of the cavitating flow. They demonstrated that their method is capable of simulating the dynamics of unsteady cavitating flow. Ji et al. (2014) investigated numerically the structure of the cavitating flow around a twisted hydrofoil using a mass transfer cavitation model and a modified RNG k-ε model with a local density correction for turbulent eddy viscosity. Cavity structures and the shedding frequency agreed fairly well with experimental observations. Goncalves and Charriere (2014) proposed an original formulation for the mass transfer between phases to study one-dimensional inviscid cavitating tube problems. Numerical results are given for various inviscid cases and unsteady sheet cavitation developing along venturi geometries and compared with experimental data. Ji et al. (2015) studied the behavior of cavities around a NACA66 hydrofoil numerically by using Large Eddy Simulation (LES) coupled with a homogeneous cavitation model. Various fundamental mechanisms governing the complex cavitating flow behaviors, including the cavitation shedding dynamic evolution, cavitation–vortex interaction and cavitation excited pressure fluctuation, were examined and summarized.

In this research, we consider cavitation and supercavitation over hemispherical head-form body and a conical cavitator using an open source package, that is, OpenFOAM. We used Kunz and Sauer mass transfer models combined with both of the LES and k-ω SST turbulence models to simulate cavitating flows. A compressive pressure form of the volume of fluid (VOF) method is employed to track the interface of liquid and vapor phases.
Additionally, we present a correlation for the cavity length and diameter for hemispherical head-form body and provide a detailed comparison of different turbulence and mass transfer models over a wide range of cavitation numbers especially in very low cavitation numbers such as $\sigma = 0.07, 0.05, 0.02$.

2. Mathematical model

2.1. Governing equations

Continuity and momentum equations for a homogeneous mixture multiphase flow are given as follow:

$$ \partial_t (\rho v) + \nabla \cdot (\rho v \otimes v) = - \nabla p + \mathbf{v} \cdot \mathbf{s}, $$

$$ \partial_t \rho + \nabla \cdot (\rho v) = 0 $$

(1)

The rate-of-strain tensor, $D$, is given as:

$$ D = \frac{1}{2} (\nabla \mathbf{v} + (\nabla \mathbf{v})^T) $$

(2)

where viscous stress tensor defined as $s = 2\mu D$.

A multiphase flow modeling should be used to describe a phase change from liquid to vapor that happens under cavitation. In this study, we consider a “two-phase” mixture method. This method uses a local vapor volume fraction transport equation together with a source term for the mass transfer rate between the two phases due to cavitation as follows:

$$ \partial_t \gamma + \nabla \cdot (\gamma \mathbf{v}) = m $$

(3)

The mixture density $\rho$ and viscosity $\mu$ are given as follows:

$$ \rho = \gamma \rho_l + (1 - \gamma) \rho_v $$

$$ \mu = \gamma \mu_l + (1 - \gamma) \mu_v $$

(4)

(5)

2.2. Mass transfer modeling

In this work, we employed two different mass transfer models Kunz and Schnerr-Sauer. Kunz et al. (2000) recommended a semi-analytical cavitation model and currently is one of the mass transfer models implemented in the OpenFOAM. This model has two different terms for vapour production and destruction:

$$ \frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \mathbf{v}) = \frac{C_{\text{dest}} \rho_l \text{Min}(\rho_l - P_B, 0) \gamma}{\rho_l \rho^l_{\infty}} + \frac{C_{\text{prod}} (1 - \gamma)^2}{\rho_l \rho^l_{\infty}} $$

(6)

The first term in the right-hand expresses vapor production that is proportional to the pressure drop from the vapor pressure and reduces the amount of liquid phase. The other term considers condensation and is proportional to the third power of the volume fraction (function $\gamma$ increases). $C_{\text{dest}}$ and $C_{\text{prod}}$ are two empirical coefficients, whose values were set to 1000 and 75 in the OpenFOAM, according to the employed discretization of the equations and so as to procure the best overall agreement with analytical relations and experimental data from various geometries (Roohi et al., 2013; Decaix and Goncalves, 2013; Roohi et al., 2016). The characteristic time (mean flow time) is defined $\tau = \frac{D_{\text{cavitation}}}{U_{\infty}}$. Kunz model reconstructs the cavity boundary quite correctly, especially in the closure region where there is a continuous flow of the re-entrant liquid jet and detachment of small vapor structures from the cavity cloud (Yu et al., 2014). Kunz’s model assumes a moderate rate of constant condensation to reconstruct this phenomenon quite accurately.

A mass transfer model was extracted by Sauer as follows (Sauer, 2000; Schnerr and Sauer, 2001):

$$ \frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \mathbf{v}) = \frac{\rho_l \rho^l_{\infty}}{\rho} \left(1 - \gamma \right)^2 \frac{3}{2} \frac{2 \rho_l - P}{\rho_l} $$

(7)

This model is a function of bubble numbers per volume unit and bubble diameter. In Eq. (7), $R_0$ is assumed to be the same for all of the bubbles given by:

$$ R_0 = \left( \frac{3}{4\pi \rho_l} \frac{1}{1 - \gamma} \right)^{1/3} $$

The final expression of the Schnerr Sauer model is written in Eq. (9).

$$ \begin{cases} \frac{m^+}{C_\text{d}} = C_\text{s} \rho_l \rho^l_{\infty} (1 - \gamma \delta) \sqrt{\frac{2 \rho_l - P}{\rho_l}} < P_B \\ \frac{m^-}{C_\text{d}} = C_\text{s} \rho_l \rho^l_{\infty} (1 - \gamma \delta) \frac{3}{2} \sqrt{\frac{2 \rho_l - P}{\rho_l}} > P_B \end{cases} $$

(9)

where $C_\text{d}$ and $C_\text{s}$ are empirical coefficients that are related to processes of condensation and vaporization, and the recommended value is 1 for both of them. In usual cases, the typical bubble size $R_0$ is $1 \times 10^{-6}$ m in water. Schnerr–Sauer model is constructed from the Rayleigh–Plesset equation and requires estimation of the initial number of bubbles ($n_0$), whose value was set as $1.6 \times 10^9$ in OpenFOAM in order to obtain a suitable agreement with experimental results or analytical relations for different geometries (Roohi et al., 2013; Decaix and Goncalves, 2013; Roohi et al., 2016). According to Eq. (8), bubble radius is a function of the local vapor volume fraction at each location.

This model ignores bubble interactions, non-spherical bubble geometries, and local mass–momentum transfer. It considered the balance of forces over spherical bubbles. This issue becomes important in predicting cavity region, especially in the case of super cavitation (Wu et al., 2005).

2.3. K-ω SST turbulence model

K-ω shear stress transport (SST) is one of the turbulence models employed in this study. Menter (1994) developed K-ω SST effectively to blend the accurate formulation of the K-ω model in the near-wall region and, K-ε model in the far field. Turbulence kinetic energy and specific dissipation rate are calculated as follows:

$$ \frac{\partial (\rho \mathbf{k})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{k}) = \nabla \cdot (\mu \nabla \mathbf{k}) + \beta^\prime \rho \mathbf{w} \mathbf{\omega} $$

$$ \frac{\partial (\rho \mathbf{\omega})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{\omega}) = \nabla \cdot (\mu \nabla \mathbf{\omega}) + \frac{\omega^2}{k} \left( \frac{\rho_1 \rho_2 \mathbf{\omega}}{\delta} \right) - \beta \rho \mathbf{\omega}^2 $$

(10)

$$ + (1 - F_1) \rho \sigma_{\omega u} \frac{1}{\sigma_{\omega u}} \frac{\partial \rho \mathbf{\omega}}{\partial \delta} $$

(11)

The model coefficients such as $\alpha_1, \beta_1, \alpha_2, \sigma_0$ are linear combinations of the corresponding coefficients of the K-ω and modified K-ε turbulence models as follows:

$$ \psi = F_1 \psi_{ke} + (1 - F_1) \psi_{ke}, \quad \alpha_1 = F_1 \alpha_1 + (1 - F_1) \alpha_2 $$

$$ k - \omega : \alpha_1 = 5/9, \quad \beta_1 = 3/40, \quad \alpha_2 = 2, \quad \sigma_0 = 2, \quad \sigma_{0u} = 2, \quad \alpha_2 = 9/100 $$

$$ k - \varepsilon : \alpha_2 = 0.44, \quad \beta_2 = 0.0828, \quad \sigma_{0u} = 1, \quad \sigma_{0u} = 1/0.856, \quad C_{\mu} = 0.09 $$

$$ \gamma_{ij} = \frac{\mu_1}{\rho} \left( \frac{2 \rho_2 - P}{3 \rho_l} \delta_{ij} \right) - \frac{2}{3} \rho \mathbf{\omega} \mathbf{\omega} $$

(13)

$$ S = \sqrt{2 \rho_l \mathbf{\omega} \mathbf{\omega}}, \quad \gamma_{ij} = \frac{1}{2} \frac{\partial \mathbf{u}_i}{\partial x_j} $$

(14)

where $S$ denotes the magnitude of the strain rate and $\gamma_{ij}$ is the strain rate tensor.
\begin{align}
F_1 &= \tanh(F^4) \\
\Gamma &= \min \left( \max \left( \frac{\sqrt{k}}{\beta^2 \omega_0^2} \frac{500 \nu}{\omega_0^2} \right), \frac{4 \rho \sigma_{\omega_2} k}{CD_{\omega_2} \omega_0^2} \right) \\
CD_{\omega_2} &= \max \left( 2 \rho \sigma_{\omega_2} \omega_0^2 \frac{1}{\omega_0^4} \partial_3 \partial_3 \Gamma \right) + 10^{-20}
\end{align}

This model mixed the Wilcox K-\omega and the Launder–Spalding K-\epsilon model advantages; however, it cannot predict the starting point and the amount of the flow separation from smooth surfaces, due to the over-prediction of the eddy-viscosity, i.e., the transport of the turbulence shear stress is not correctly taken into account. Adding a limiter to the formulation of the eddy-viscosity, an appropriate transport behavior could be obtained:

\begin{equation}
\mu_t = \rho \frac{\mu}{\max(\mu, S^2 F_2)}
\end{equation}

where \( S \) is an invariant measure of the strain rate, and \( F_2 \) is a blending function as follows:

\begin{align}
F_2 &= \tanh(F_2^2) \\
F_2^2 &= \max \left( 2 \sqrt{k} \frac{500 \nu}{\omega_0^2} \right)
\end{align}

When the underlying assumptions are not correct for free shear flow, \( F_2 \) restricts the limiter to the wall boundary layer.

Following Menter et al. (2003), OpenFOAM uses a blending function depending on \( y^+ \) for the near-wall treatment. The solution for \( \omega \) in the linear and the logarithmic near-wall region are:

\begin{equation}
\omega_{\log} = \frac{1}{0.33} \frac{\mu_t}{y^+ \nu} \omega_{\nu} = \frac{6 \nu}{0.075 y^+}
\end{equation}

The above equations are rewritten in terms of \( y^+ \) to deduce a smooth blending function:

\begin{equation}
\omega_t(y^+) = \sqrt{\omega_{\nu} \omega_{\log}(y^+)} + \omega_{\log}(y^+)
\end{equation}

A similar formulation is used for the velocity near the wall:

\begin{align}
\frac{u_t^v}{y^+} &= \frac{U_1}{y^+} + \frac{U_1}{p_1 h(y^+) + c} \\
\frac{u_t^w}{y^+} &= \frac{U_1}{y^+} + \frac{U_1}{p_1 h(y^+) + c}
\end{align}

This formulation expresses the relation between the velocity near the wall and the wall shear stress. A zero flux boundary condition is applied for the \( k \)-equation, which is correct for both the low-Re and the logarithmic limit. This model exploits the robust near wall formulation of the underlying k-\omega model and switches automatically from a low-Reynolds number formulation to a wall function treatment based on the grid density (Menter et al., 2003).

2.4. Large eddy simulation

Large eddy simulation (LES) turbulence approach is based on computing the large, energy-containing structures that are determined on the computational grid, while the smaller sub-grid eddies are modeled. LES equations are theoretically derived from Eq. (1) (Fureby and Grinstein, 2002). In LES, all variables, i.e., \( f \), are split into grid scale (GS) and subgrid scale (SGS) components, i.e., \( \tilde{f} = G \ast f \) is the GS component, \( G = G(\Delta x) \) is the filter function, and \( \Delta = \Delta(x) \) is the filter width (Ghosal, 1996). The LES equations can be expressed as:

\begin{equation}
\partial_t (\rho \overline{f}) + \nabla \cdot (\rho \overline{f} \times \nabla) = - \nabla \overline{P} + \nabla \cdot \left[ \left( \sigma - B \right) \right] + \partial_t (\rho \overline{\nu} \nabla \overline{u}) = 0
\end{equation}

where the rate-of-strain tensor is expressed as \( D = \frac{1}{2} \left( \nabla \overline{u} + \nabla \overline{u}^T \right) \) and unresolved transport term \( B \) can be exactly decomposed as (Bensow and Fureby, 2007):

\begin{equation}
B = \rho \overline{\nu} \left( \overline{\nabla \times \nabla} - \overline{\nabla} \overline{\nabla} \overline{\nu} + B \right)
\end{equation}

where \( B \) needs to be modeled. The most common subgrid modeling approaches utilize an eddy or subgrid viscosity, \( \nu_{SGS} \), where \( \nu_{SGS} \) can be computed with a wide variety of methods. In eddy-viscosity models,

\begin{equation}
B = \frac{2}{3} \nu T k - 2 \nu \Delta^{2} \overline{\nu}
\end{equation}

In the current study, sub-grid scale terms are modeled using “one equation eddy viscosity” model. It should be noted once we compared the most popularly employed SGS model, i.e., standard “Smagorinsky” with “one equation eddy viscosity” in many aspects such as the computational costs and cavity cloud shape. It is observed that computational costs could decrease up to 30%; however, one important point should be noted: as there are not any experimental pictures of the cavity shape to compare the vapor shedding, fluctuating cavity behavior, or re-entrant jet with numerical solutions obtained from one equation eddy viscosity model or Smagorinsky SGS model, it is appropriate to use a more accurate SGC approach even though it requires higher computational costs. However, if the aim of computation is to compare general properties of the cavity cloud such as length, diameter, and pressure drag coefficient, the simulation should be performed with less expensive SGS models as these parameters are insensitive to the employed SGS model. In order to obtain \( k \), one-equation eddy viscosity model (OEEVM) uses the following equation:

\begin{equation}
\partial_t (\rho k) + \nabla \cdot (\rho \overline{\nu} \nabla k) = - \nabla \cdot (\overline{\nu} \nabla \overline{\nu}) + \overline{\rho \nu \delta}
\end{equation}

\begin{equation}
\varepsilon = C_\nu \frac{k^{3/2}}{\Delta}
\end{equation}

\begin{equation}
\mu_k = C_{\nu} \Delta \frac{k}{\sqrt{k}}
\end{equation}

where \( C_\nu \) and \( C_k \) are set as 1.048 and 0.094 in OpenFOAM, respectively.

A blending function is employed in the OpenFOAM’s LES to manage different values of \( y^+ \) for the first grid cell and its influence in near-wall function selection (De Villiers, 2006; Damiána and Nigro, 2010). The blending function is given by Spalding Law as follows (Spalding, 1961):

\begin{equation}
y^+ = u^+ \frac{1}{F} \left[ e^{e^{-y^+} \left( e^{2y^+} - 1 - e^{-y^+} \left( e^{2y^+} - 1 \right) \right)} \right]
\end{equation}

where \( \kappa = 0.4187 \), \( E = 9 \), \( y^+ = y^{+1} u^+ / \nu \) and \( u^+ = \overline{u} / \nu \). The principal advantage of using such a unified wall function is that the first off-the-wall grid point can be located in the buffer or viscous regions \( y^+ < 30 \) without the loss of accuracy inherent in the logarithmic profiles limited validity (De Villiers, 2006).

2.5. Volume of fluid model

The volume of fluid (VOF) method is adapted to reconstruct the interface between the liquid and vapor phases. \( \gamma \) is the volume fraction of fluid 1 determined in the following form:

\begin{equation}
\gamma_1 = \begin{cases} 
1 & \text{fluid 1(Liquid)} \\
0 & \text{fluid 2(Vapor)}
\end{cases}
\quad 0 < \gamma < 1 \quad \text{at the interface}
\end{equation}
OpenFOAM uses the following compressive-velocity ($v_c$) form of the VOF equation:

$$\frac{\partial \gamma}{\partial t} + V \cdot \nabla \gamma + \nabla \cdot \left[ V_c \gamma (1 - \gamma) \right] = \dot{m}$$ (33)

where $\dot{m}$ is given by the selected mass transfer model (Kunz and Sauer), $v_c$ is the compressive velocity term suggested in Rusche (2002). The additional term in Eq. (33) is only active at the interface region and acts as a surface compression term. It improves the interface resolution and limits the smearing of the interface. For more details about compressive velocity VOF approach, see Refs. (Roohi et al., 2013; Klostermann et al., 2013).

3. Numerical method

3.1. OpenFOAM validation

Before simulating the cavitating flow behind a disk, we evaluate the accuracy of the OpenFOAM package in simulating an incompressible, turbulent, non-cavitating flow. Experimental investigation of unsteady flow behavior and vortex shedding behind a non-cavitating disk at various Reynolds numbers were reported in Miau et al. (1997). Here, we set the Reynolds number equal to $Re = 2.8 \times 10^4$. Fig. 1 shows the computational domain around the disk with a diameter of 0.1 m. This value is chosen in accordance with the geometrical data of the water tunnel used in Miau et al. (1997). For a typical time step, Fig. 2 compares the experimental and numerical solution for normalized velocity contours taken through the mid-plane of the disk. As the figure shows, suitable agreement is observed, especially for velocity contours behind the disk for LES turbulence model. Compared to RANS, LES is a more viable tool for predicting unsteady flows. The LES approach has become increasingly popular in the CFD application because of this unique nature. Using RANS models, cavitating flows often exhibit significant unsteadiness and therefore require considerable resolution in time and space. In addition, the predicted Strouhal number for LES was around 0.14, which is within 5.4% of the experimental value (Miau et al., 1997) reported at this Reynolds number.

3.2. Solution domain and boundary conditions

Fig. 3-a shows the 3D computational domain of the hemispherical head-form body with the diameter of $D = 0.2$ m. It has the following geometrical parameters: the computational domain is $10D$ in length: body length of $80D$ and distance between the domain inlet and the head of the body is $20D$, and $40D$ in height.

Fig. 3-b shows the 3D computational domain of cone cavitator with the diameter of $D = 0.8284$ m and the height of $H = 1$ m. The computational domain is $120D$ in length (the length behind the cavitator is $100D$ and distance in front of the cavitator is $20D$) and $52D$ in height. The boundary conditions of both cases are similar to each other as illustrated in Fig. 3. Two non-dimensional numbers: Reynolds number ($Re = \rho U_\infty D / \mu$) and cavitation number ($\sigma = (P_{\infty} - P_v) / 0.5 \rho U_\infty^2$) are considered. Reynolds numbers for hemispherical head-form body and conical cavitator are considered $4.4 \times 10^6$ and $1.8 \times 10^7$, respectively.

3.3. Computational mesh and time discrimination

3.3.1. Grid dependency study

As Figs. 4 and 5 illustrate, we used structured quadrilateral meshes for both cases in this paper. We considered the geometry as three-dimensional although both of geometries are axisymmetric, and the inflow conditions are steady; however, the flow is three-dimensional and unsteady due to the vortex shedding behind the body at the investigated Reynolds/cavitation numbers. Accuracy of results strongly depends on the mesh size near the body and especially around the cavity closure regions. In the other
word, since the interaction between the near body flow and cavity is crucial; the mesh near the wall of the test body should be well refined. Fig. 4-c shows that there is a dense mesh at the length of $L = 40D$, $25D$, $15D$ for cavitation numbers $\sigma = 0.02$, $\sigma = 0.05$ and $\sigma > 0.1$, respectively. The grid size is progressively increased in the other regions, where the variations of flow proportion are small. This method helps to save the computational cost and decrease the total cell numbers.

Table 1 compares simulation results for conical cavity parameters (length and diameter) obtained using different grid sizes with the analytical data. The grid used for $\sigma > 0.07$ cases are relatively coarse compared to those used for $\sigma = 0.02$. From Table 1, it is observed that Grid 2 (which is $1 \times 10^6$ cells for $\sigma > 0.07$ and $2.4 \times 10^6$ cells for $\sigma = 0.02$) provides close solutions for predicting different characteristic of the cavity behind the conical cavitator. Therefore, we performed our simulations using Grid 2. For hemispherical head-form cylinder, we performed our simulations using $2.4 \times 10^6$ cells for $\sigma = 0.02$ and $1.5 \times 10^6$ cells for $\sigma > 0.05$. The maximum values of $y^+$ were 241 and 275, and the mean values of $y^+$ were 8.59 and 7.9 for $\sigma = 0.02$ in conical cavitator case and hemispherical head-form cylinder case, respectively. For $30 < y^+ < 300$ at most positions, i.e., adjacent to the body region, OpenFOAM uses wall functions.

Table 2 compares the required time to reach steady state condition for different models for two cavitation numbers for the cone cavitator. Simulations were performed in parallel using 4 and 12 cores of Intel Core™ i7-2600K CPU equipped with 16 GB memory RAM in $\sigma = 0.07$ and 0.02, respectively. In these simulations, calculation is performed in a manner that Courant number in the computational domain does not surpass 0.175 for $\sigma = 0.07$ and 0.09 for $\sigma = 0.02$.

### 3.4. Solution convergence and procedure

Fig. 6 shows the residuals convergence history of the liquid volume fraction (alpha) and pressure ($p$) for the five last time steps for a typical test case at $\sigma = 0.07$. The iterative convergence of unsteady flow problems depends on time step and number of iterations per time step. Residual of each parameter is defined as the normalized difference between the current and previous value of that parameter. Residuals increase at the beginning of each time step and then drop by two to three orders of magnitude.

In this work, we use PIMPLE algorithm. It is a merged PISO-SIMPLE algorithm for solving the pressure–velocity coupling. This algorithm, as shown in Fig. 7, enables a more robust pressure–velocity coupling by coupling a SIMPLE outer-corrector loop with a PISO inner-corrector loop. PIMPLE shows a better numerical stability for larger time-steps or higher Courant numbers or larger time-steps. Typically PISO and SIMPLE iterations...
Table 1
Effect of different grid sizes on the cavity length/cavitator diameter and cavity diameter/cavitator diameter behind the conical cavitator.

<table>
<thead>
<tr>
<th>Mesh for ( \sigma ) &gt; 0.07</th>
<th>Number of cells (( \times 10^6 ))</th>
<th>( \frac{L_{cavity}}{d_{cavitator}} )</th>
<th>( \frac{D_{cavity}}{d_{cavitator}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid1</td>
<td>0.75</td>
<td>15.21</td>
<td>2.20</td>
</tr>
<tr>
<td>Grid2</td>
<td>1</td>
<td>12.20</td>
<td>1.96</td>
</tr>
<tr>
<td>Grid3</td>
<td>1.5</td>
<td>13.01</td>
<td>2.20</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mesh for ( \sigma = 0.02 )</th>
<th>Number of cells (( \times 10^6 ))</th>
<th>( \frac{L_{cavity}}{d_{cavitator}} )</th>
<th>( \frac{D_{cavity}}{d_{cavitator}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid1</td>
<td>1.8</td>
<td>67.25</td>
<td>4.30</td>
</tr>
<tr>
<td>Grid2</td>
<td>2.4</td>
<td>61.46</td>
<td>3.91</td>
</tr>
<tr>
<td>Grid3</td>
<td>3</td>
<td>59.42</td>
<td>3.92</td>
</tr>
</tbody>
</table>

Fig. 4. The computational structured meshes around hemispherical head-form body, figures show close-up view of the mesh near the body and refined mesh at different locations around the body.
Table 2
Details of the computational cost of investigating test cases.

<table>
<thead>
<tr>
<th>Turbulence and mass model</th>
<th>Steady time (ms)</th>
<th>Run time (h)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma=0.07$</td>
<td>k-$\omega$ SST/Sauer</td>
<td>184</td>
</tr>
<tr>
<td>Courant=0.175</td>
<td>k-$\omega$ SST/Kunz</td>
<td>180</td>
</tr>
<tr>
<td>Grid2</td>
<td>LES/Sauer</td>
<td>120</td>
</tr>
<tr>
<td>Core numbers=4</td>
<td>LES/Kunz</td>
<td>104</td>
</tr>
<tr>
<td>$\sigma=0.02$</td>
<td>k-$\omega$ SST/Sauer</td>
<td>450</td>
</tr>
<tr>
<td>Courant=0.09</td>
<td>Grid2</td>
<td>433</td>
</tr>
<tr>
<td>Core numbers=12</td>
<td>LES/Sauer</td>
<td></td>
</tr>
</tbody>
</table>

Fig. 5. The computational structured meshes around conical cavitation figures show close-up view of the mesh near the body and refined mesh at different locations around the body.
Fig. 6. Convergence of residuals for five last time steps at $\sigma = 0.07$.

Fig. 7. PIMPLE flowchart.
are required per time step. Here, we employed two PISO iterations and one SIMPLE iteration.

4. Results and discussion

4.1. Hemispherical head-form body

Fig. 8 illustrates 3D views of isosurfaces of the volume fraction contours of the cavitating flow over hemispherical head-form body over a broad range of cavitation number from $\sigma = 0.5$ to $\sigma = 0.02$, where the latter is reported for the first time in the open literature. The k-\(\omega\) SST turbulence model with Sauer mass transfer model were employed. A considerable increase in the cavity length is observed at the lowest cavitation number. Frames in Fig. 8 indicate that the growth of the cavity length could be obtained from an inverse power law relation of the cavitation number. At $\sigma = 0.02$, cavity has a fluctuating shape and shows an unsteady behavior while the cavity shape is steady, regular and smooth at the other cavitation numbers.

Fig. 8. 3D cavitation growth over hemispherical head-form cylinder, k-\(\omega\) SST/Sauer models.

Fig. 9. Comparisons of pressure coefficients with the experimental data (May, 1975).

Fig. 10. Comparison of different turbulence and mass transfer models.
The pressure distribution, $c_p=(p-p_0)/0.5\rho U_{in}^2$, on the hemispherical head-form body surface for various cavitation numbers ($0.02 \leq \sigma \leq 0.5$) from numerical solutions (k-ω SST/Sauer, k-ω SST/Kunz, LES/Sauer, LES/Kunz) is compared with reported experimental data (Rouse and McNown, 1948) as shown in Figs. 9–11. The horizontal coordinate in these figures is the cavity length from the nose of the sphere non-dimensionalized by the radius of the cylinder. The simulated results agree well with experimental data. It confirms that the models and numerical procedures used in this paper are accurate for cavitation simulations. The pressure overshoot at the cavity closure is more prominent in the computational results as the cavitation number decreases. Fig. 9 shows a general validation of the employed schemes compared with experimental data.

Fig. 10 shows the pressure coefficient distribution for various cavitation models at $\sigma=0.1$ and 0.2. Kunz model shows less sensitivity to the turbulence model, but Sauer model strongly interacts with the turbulence model. The maximum value of $c_p$ in k-ω SST/Sauer model has higher amount than other models. However, these models do not show a significant difference with each other.

Fig. 11 illustrates $c_p$ value over a broad range of cavitation numbers with k-ω SST and Sauer cavitation model. An important point that can be found, is that a maximum overshoot in $c_p$ distribution is brightly observed at different cavitation numbers. The level of overshoot could be a function of the grid size, numerical discretization scheme, turbulence model, and cavitation model.

Figs. 12–14 illustrate detailed features of the cavitation flow around the hemispherical head-form body. Fig. 12 depicts the volume fraction of the liquid phase for different turbulence and mass transfer models from $0.02 \leq \sigma \leq 0.5$. As soon as the cavity cloud reaches its fully developed condition, it shows a periodic behavior. The re-entrant jet is generated at the cavity closure. The re-entrant jet formed due to the adverse pressure gradient and shear stress. We consider the re-entrant jet in terms of two aspects: length and strength. LES predicts a longer length for the re-entrant jet, but k-ω SST model predicts a weak re-entrant jet in all cases, an exception is $\sigma=0.1$ case, where the re-entrant jet for two models are approximately the same. However, it should be mentioned that turbulence and mass transfer models simultaneously affect on the re-entrant jet length and strength. As the cavitation number decreases, re-entrant jet becomes weaker. For the LES/Sauer model, the length of the re-entrant jet, non-dimensionalized by the cavity length ($\delta L$) at different cavitation numbers of $\sigma=0.2, 0.1, 0.05$ are 0.356, 0.331, 0.247 and 0.126, respectively. Frames (h–i) in Fig. 12 show the boundary-layer separation in the re-entrant jet region in the LES model. The separation point location is indicated by point C. For instance, separation point position is located at $(X/R)=12.88$ in LES/Sauer solution. These observations demonstrate that selection of an appropriate turbulence model is a crucial issue in the correct prediction of the cavitation physics.

Fig. 13 illustrates $c_p$ contours for the same cases that were reported in Fig. 12. The pressure inside the cavity are observed to stay close to the vapor pressure, giving an absolute value of $c_p$ approximately equal to the cavitation number $\sigma$. $c_p$ distribution of each model has roughly the same behavior inside the cavity cloud at different cavitation numbers. The value of $c_p$ at the cavity closure region with k-ω SST and Sauer are $c_p=0.41, 0.43, 0.28, 0.22$ at $\sigma=0.2, 0.1, 0.05, 0.02$, respectively. The maximum overshoot in $c_p$ distribution is observed at $\sigma=0.1$.

Fig. 14 shows velocity contours and streamlines for the same cases that reported in Fig. 12. In LES/Sauer solutions (right frames), the vortices are distributed from a single point as indicated by an arrow in each frame. On the left of the arrow, the flow direction is reversed from the streamwise direction; however, in the k-ω SST/Sauer solution, there are either two separate vortices ($\sigma=0.2$) or one elongated vortex. The vortex strength is higher in k-ω SST/Sauer solution.

Fig. 15 shows the growth of the cavity length at different cavitation regimes. The $L/R$ values obtained from two numerical approaches (k-ω SST/Sauer, LES/Sauer) agree with each other. The evolution of cavitation can be described in three main stages: (1) the flow speed is relatively high over the cylinder body. Once the pressure falls below the saturation vapor pressure, water begins to vaporize. Then, the cavity flows and grows gradually. (2) Re-entrant jet appears and flow moves back to the forepart of the body and cause the primary cavity to separate from the body. Then, the bubbles shedding start and bubbles moves towards the tail of the body. Cavity cloud becomes smaller. (3) New vaporous cavity region starts growing again (after $t > 440$ ms). This is the fluctuating characteristic of the cavitation evolution.

Figs. 16 and 17 show the cavity length and diameter at different cavitation numbers. The parameters are normalized by the cylinder diameter. Based on the current numerical solutions, we deduce inverse power law correlations for the cavity length and diameter over the hemispherical body as follows:

$$\frac{L}{R} = \frac{0.28}{\sigma^{0.28}} (0.02 \leq \sigma \leq 0.5)$$
$$\frac{d}{R} = \frac{0.8}{\sigma^{0.8}} (0.02 \leq \sigma \leq 0.5)$$

The cavity length and diameter exponentially increase with decreasing the cavitation number. The results of different models are quite close to each other.

4.2. Conical cavitation

In this section, we consider cavitation behind a conical cavitor. Fig. 18 shows 3D view of the cavity cloud behind a conical cavator at cavitation number $\sigma=0.07$ obtained using different turbulence and mass transfer models. All models show nearly the same length and diameter for the cavity clouds. Fig. 19 illustrate 3D view of the cavitating flow in $\sigma=0.02$. The cavity has a roughly...
Fig. 12. Volume fraction contours for different turbulence and mass transfer models at $0.02 \leq \sigma \leq 0.5$. 

- $k-\omega$ SST, Sauer $\sigma=0.5$
- LES, Sauer $\sigma=0.5$
- $k-\omega$ SST, Sauer $\sigma=0.2$
- LES, Sauer $\sigma=0.2$
- $k-\omega$ SST, Sauer $\sigma=0.05$
- LES, Sauer $\sigma=0.05$
- $k-\omega$ SST, Sauer $\sigma=0.02$
- LES, Sauer $\sigma=0.02$
Fig. 12. (continued)
Fig. 13. Pressure coefficient contour for different mass transfer/turbulence models $0.02 \leq \sigma \leq 0.5$. 

- **k-ω SST, Sauer $\sigma=0.5$**
- **LES, Sauer $\sigma=0.5$**
- **k-ω SST, Sauer $\sigma=0.2$**
- **LES, Sauer $\sigma=0.2$**
- **k-ω SST, Sauer $\sigma=0.1$**
- **LES, Sauer $\sigma=0.1$**
- **k-ω SST, Sauer $\sigma=0.05$**
- **LES, Sauer $\sigma=0.05$**
- **k-ω SST, Sauer $\sigma=0.02$**
- **LES, Sauer $\sigma=0.02$**
pulsating surface in the k-ω SST/Sauer solution while LES/Sauer predicts smooth and regular shape. Two models show a steady behavior with a very long cloud length at σ=0.02. The movie of cavitation growth at σ=0.02 is provided as the supplementary movie of this paper.

Supplementary material related to this article can be found online at http://dx.doi.org/10.1016/j.oceaneng.2015.12.010.

The contours of volume fraction are illustrated in Fig. 20 as a 2-D section in the z plane (z-axis is shown in Fig. 18) at different cavitation numbers from various mass transfer and turbulence models. All models show nearly the same length and diameter for the cavity clouds. A sharp interface is visible in the cavity domain which is due to using an accurate VOF method. The re-entrant jet in the k-ω SST model is weaker than the LES model. It is because of that the k-ω SST model fails to properly predict the onset and amount of the flow separation and re-entrant jet from smooth surfaces with suitable details, due to the over-prediction of the eddy-viscosity, i.e., the transport of the turbulent shear stress is not properly taken into account (Baradaran Fard and Nikseresht, 2012). In LES model, however, separation of scales within the flow is accomplished by a low-pass filtering of the Navier–Stokes Equations. In contrast with RANS approaches, which are based on solving for an ensemble average of the flow, LES naturally and consistently allows for medium- to small-scale, transfer. This is
an important property in order to be able to capture the mechanisms governing the dynamics of the cavity such as re-entrant jet. LES/Sauer combination indicates a sharp re-entrant jet that is stronger compared to the LES/Kunz model. Therefore, cavity closure region is unsteady in this model in comparison with other models. K-ω SST/Kunz solutions show a rough cavity boundary with a weak re-entrant jet. The frames in Fig. 20 indicate that re-entrant weakens by decreasing the cavitation number.

Fig. 21 illustrates contours of $c_p$ for the same cases reported in Fig. 20. Ahead of the cone, pressure increases due to the formation of the frontal stagnation point. Behind the body, the flow separates at the sharp edges of the cone and the resultant drop in pressure creates a vaporous cavity region.

Fig. 22 shows velocity contours with streamline for the same cases that reported in Fig. 20. Evidently, combination of turbulence and mass transfer models determines the streamline and strength differently. Vortex strength or circulation, defined as $\int \omega \cdot dA$, i.e., product of the vorticity normal to the surface enclosed by the cavity region is higher in the LES/Sauer solution. In LES/Sauer model, vortices are distributed from a division point. In k-ω SST/Kunz solution, there are two elongated vortex behind the cavitator, but at $\sigma=0.2$, there are four strong vortices behind the cavitator and in the cavity closure region.

Figs. 23 and 24 show predicted maximum cavity length and diameter non-dimensionalized by the cavitator diameter. We evaluate our result with an extensive set of Reichardt, Garabedian and Logvinovich (May, 1975) analytical relations. The set of analytical relations are given as follows:

Reichardt’s relations: This relation relates the steady cavity length to the cavitation number:

$$L = \sigma + 0.008 \ln \left(1/\sigma + 0.008\right)$$

Reichardt also derived a theoretical formula for the cavity diameter that is a function of the drag coefficient and cavitation number.

$$d = \sqrt{\frac{C_d}{\sigma^2 (1 - 0.132\sigma^{0.5})}}$$

Also he presented the drag coefficient as follows:

$$C_d = C_{d0}(1 + \sigma)$$

where $C_{d0}$ is considered as 0.30275 for conical cavitators.

Garabedian relations: Garabedian obtained asymptotic relations for main properties of the cavities past cavitators as follows:

$$L = \sqrt{\frac{C_d \ln \sigma}{\sigma}}$$

$$D = \sqrt{\frac{C_d}{\sigma}}$$

Logvinovich relations: Logvinovich derived formulas for properties of the cavities depending on the cavitation number as
follows:

\[
\frac{L}{D_n} = \frac{1.1}{\sigma + \frac{4(1-2\alpha)}{1+144\alpha^2}} \sqrt{\frac{c_x}{\kappa} \ln \frac{1}{\sigma}}
\]  

(41)

\[
\frac{D}{D_n} = \sqrt{\frac{c_x}{\kappa \sigma}}
\]  

(42)

\[
\kappa = \frac{1 + 50\sigma}{1 + 58.2\sigma}
\]  

(43)

\[
\begin{align*}
\alpha & = \alpha(0.915 + 9.5\alpha), & 0 < \alpha < \frac{1}{2} \\
\end{align*}
\]  

(45)

where \( \alpha \) is the cone half-angle. It should be noted that above-mentioned analytical relations were derived based on the potential flow theories. A general agreement between numerical approaches and analytical relation is observed, especially at lower cavitation numbers. However, the figures indicate that there is deviation between analytical relations and numerical simulations at higher cavitation numbers. It is expected as the analytical relations were derived from potential flow theory assuming fully developed cavitation.

5. Conclusion

In this work, we investigated cavitation and supercavitation around 3D hemispherical head-form body and a conical cavitator over a wide range of cavitation numbers. A compressive VOF method was employed to track the interface between liquid and vapor phases. Benefiting from large eddy simulation (LES) and k-ω SST turbulence models as well as Kunz and Sauer mass transfer models, we simulated cavitation accurately. Simulation is performed under the framework of the OpenFOAM package. One of the main contributions in this work is to present a correlation between the cavity length and diameter for hemispherical head-
form body. Secondly, we provided a detailed comparison of different turbulence and mass transfer models in very low cavitation numbers such as \( \sigma = 0.07, 0.05, 0.02 \) for both considered cases. Our numerical results are compared with the experimental data for hemispherical head-form body and an extensive set of analytical relations for the conical cavity characteristics such as cavity length and diameter. Combined models (k-\( \omega \) SST/Sauer, k-\( \omega \) SST/Kunz, LES/Sauer, LES/Kunz) predict approximately close results for general parameters of the cavity such as non-dimensional cavity length and diameter. However, we showed that these models differ in predicting the structure of the re-entrant jet, distribution of pressure in the flow field, velocity contours and vortices in the

**Fig. 20.** Volume fraction contours for the conical cavitator in a broad range of cavitation numbers.
Fig. 21. Contour of pressure in different cavitation numbers.

Fig. 22. Contour of velocity with streamline.
cavity cloud, boundary layer separation, as well as cavity cloud shape.

References


Fig. 23. Comparison of the numerical cavity length/cavitator diameter with analytical data.

Fig. 24. Comparison of the numerical cavity diameter/cavitator diameter with analytical data.