



Research paper

Investigation of numerical solution approaches for the cavitating flow analysis of twisted hydrofoils

Sinem Öksüz^{a,c,*}, Onur Usta^b, Fahri Celik^c^a Department of Marine Engineering, Faculty of Maritime, Piri Reis University, 34940, Istanbul, Turkey^b Department of Naval Architecture and Marine Engineering, Turkish Naval Academy, National Defence University, 34942, Tuzla, Istanbul, Turkey^c Department of Naval Architecture and Marine Engineering, Faculty of Naval Architecture and Maritime, Yıldız Technical University, 34349, Istanbul, Turkey

ARTICLE INFO

Keywords:

Twisted hydrofoil
Cavitation development
Cavitation cycle
Delft 11 hydrofoil
NACA0009 and NACA0015
RANS
DES

ABSTRACT

In this study, geometrical parameters such as twist, section thickness, and angle of attack are numerically investigated for their influence on lift and drag coefficients, as well as cavitation development. The Delft 11 Hydrofoil, a twisted version of the NACA0009 foil with a -2° angle of attack, is used as the benchmark geometry for validation studies under both non-cavitating and cavitating flow conditions. After the validation studies, the geometries of the NACA0009 and NACA0015 hydrofoils were redesigned as half-twisted and twisted. These redesigned hydrofoils, along with the original no-twist versions, were analyzed under cavitating flow conditions within an angle of attack range of $-2^\circ < \text{AoA} < 5^\circ$. In the study, three-dimensional, unsteady, cavitating flow is modelled by two different solution approaches, RANS and DES, with the SST Menter k- ω turbulence model. The Schnerr-Sauer cavitation model is employed to obtain cavitation formation. The same physical conditions with RANS simulations are performed using a refined mesh with the DES method, focusing particularly on accurately and precisely obtaining cavitation development, especially cavitation cycles. Additionally, DES analyses were performed for various time step values, demonstrating the specific influence of the time step on cavitation cycles. The results show that while the RANS method effectively predicts the lift and drag forces of the hydrofoil, the DES method is crucial for capturing cavitation dynamics with greater precision, particularly for obtaining cavitation cycles. The results also indicate that twisted hydrofoil geometries produce more lift and drag force than half-twisted and no-twist geometries for both NACA0009 and NACA0015 hydrofoils. In addition, the cavitation formation around the hydrofoil generally increases with the twistiness.

1. Introduction

Hydrofoils have typical applications in the maritime industry such as rudders, propeller blades, turbines, submarines. Due to their extensive applicability, hydrofoils remain a subject of ongoing research and advancement within the field of marine engineering.

Twisted hydrofoils represent a distinctive design characterized by a curved alteration along their span, which offers advantages in hydrodynamic efficiency and stability. Hydrofoil blades generate lift as they interact with water flow, and their specific shape significantly influences the lift they produce. Introducing a twist to the hydrofoil means that the angle at which water strikes the wing (known as the angle of attack, measured between the wing's chord line and the oncoming flow) varies along the wing's span. This concept draws inspiration from the way birds' wings naturally flex to adapt to various flight conditions (Liu

et al., 2006).

Twisted hydrofoils offer several benefits. By adjusting the angle of attack along the wing span, it becomes possible to minimize drag and decrease energy losses, particularly when operating at varying speeds. Secondly, they enhance stability by achieving a more controlled lift distribution. This can result in smoother manoeuvres and improved handling, especially when dealing with changing flow conditions. Another advantage is the efficient distribution of lift along the foil's geometry. In hydrofoil systems, maintaining an optimal lift distribution across the foils' span is crucial, and twisted hydrofoils contribute to achieving a balanced and efficient lift profile. Lastly, the curved modification enhances manoeuvrability. The ability to fine-tune lift distribution across the span enhances the hydrofoil's manoeuvring capabilities, offering better control during turns and other manoeuvres (Kim et al., 2014).

* Corresponding author. Department of Marine Engineering, Faculty of Maritime, Piri Reis University, 34940, Istanbul, Turkey.

E-mail address: soksuz@pirireis.edu.tr (S. Öksüz).

<https://doi.org/10.1016/j.oceaneng.2024.119198>

Received 3 July 2024; Received in revised form 19 August 2024; Accepted 4 September 2024

Available online 23 September 2024

0029-8018/© 2024 Elsevier Ltd. All rights are reserved, including those for text and data mining, AI training, and similar technologies.

Due to these advantages, twisted foils find application in a range of vessels including boats, catamarans, and trimarans. Designers can customize the twist along the span of these foils to achieve various benefits such as drag reduction, optimized lift distribution for enhanced stability, and decreased fuel consumption across different speeds and sea conditions. Another significant domain where twisted hydrofoils play a crucial role is in submarines, serving various purposes like depth control, manoeuvrability, and resistance reduction (Soto and Bhattacharya, 2023).

Cavitation poses a significant challenge for hydrofoils, resulting in surface pitting, erosion, and potential structural breakdown of affected parts, ultimately necessitating expensive repairs and operational halts. In addition to physical damage, cavitation diminishes equipment effectiveness, raises energy consumption, and generates noise pollution (Usta and Korkut, 2019).

In numerical modelling of cavitation, achieving precise turbulence modelling and selecting suitable cavitation models are essential for the accurate prediction of the complicated, multi scale and multi-phase dynamics characterizing these flows (Köksal et al., 2021; Usta and Korkut, 2018).

The RANS approach averages turbulent characteristics across all scales, thus inadequately capturing the finer details and dynamic aspects of cavitation. Nonetheless, several studies (Li et al., 2009; Liu and Hong, 2010; Li, 2012; Paik et al., 2013; Liu et al., 2014; Ji et al., 2014; Melissaris et al., 2019; Shi et al., 2021; Kumar et al., 2022) have employed combinations of turbulence models like SST $k-\omega$ or RNG $k-\epsilon$ along with cavitation models such as Schnerr and Sauer, Zwart or Singhal to simulate cavitation phenomena.

Ji et al. (2015) employed the Large Eddy Simulation (LES) methodology to simulate the development of cavitation around a NACA66 hydrofoil, with specific emphasis on examining cavitation structures and the dynamics of shedding. They found that pressure fluctuations primarily originated from cavity volume acceleration. Wu et al. (2016) conducted LES to examine the impact of grid density on time-dependent cavitating flow around a twisted wing. Their findings demonstrated that a finer grid resolution significantly enhanced the precision of predictions for cavitating flow. Vaz et al. (2017) conducted a comparative analysis of three methodologies for predicting cavitation on a three-dimensional hydrofoil: pure Reynolds-Averaged Navier-Stokes (RANS), RANS with a Rebound correction for eddy viscosity, and Delayed Detached Eddy Simulation (DDES), Scale-Resolving Simulation. Their investigation revealed that the DDES approach, while being more efficient and offering improved dynamics, exhibited slightly lower accuracy in predicting cavity shedding compared to the other two methods. In their study, Long et al. (2018) employed the LES technique along with the Zwart cavitation model to simulate the dynamics of transient cavitating turbulent flow around a three-dimensional hydrofoil. Through Lagrangian analysis, it was observed that the interplay of mainstream, re-entrant, and side-entrant jets played a significant role in both primary and secondary shedding within cavitating flow. In their study, Zhang et al. (2018) employed a modified FBM turbulence model and a homogeneous cavitation model to investigate the shedding of cavities around a three-dimensional twisted hydrofoil. Their findings pinpointed the primary cause of cloud cavity shedding as the adverse pressure gradient-induced re-entrant jet, with secondary shedding resulting from the interaction between the re-entrant jet and the side-entrant jet. Movahedian et al. (2019) conducted an investigation into the dynamics of cavitating flow around a 3-D twisted NACA 16012 hydrofoil, utilizing the LES method and the Volume of Fluid (VOF) approach. They examined the presence of side-entrant jets in detail. Their findings revealed that the pressure coefficients on the hydrofoil exhibited an increase as a result of the downstream collapse of the horseshoe-shaped vortex. Chen et al. (2019) applied compressible LES for the purpose of simulating cavitating flow around a NACA66 hydrofoil. In another study, Cheng et al. (2021) conducted an investigation into the impact of tip leakage on cavitating flow over a NACA0009 hydrofoil. Their study also employed

the LES method in combination with the Schnerr–Sauer cavitation model. Melissaris et al. (2019) simulated cavitating flow region around Delft hydrofoil by using URANS method by designing appropriate mesh configuration by comparing analyse results with the experimental data. They also correlated the cavitation development and pressure with cavitation erosion and proposed numerical models to predict eroded region on the hydrofoil. Arabnejad et al. (2020) conducted a comprehensive study on the dynamics of leading-edge cavitation and the resulting cavitation erosion on a NACA0009 hydrofoil. Their investigation involved a combination of experimental and numerical methodologies. The researchers identified four specific regions on the hydrofoil susceptible to high-risk erosion. Furthermore, their findings indicated that small-scale structures resulting from secondary shedding exhibited greater erosive potential compared to larger-scale cloud cavities. Pant and Frankel (2021), have utilized a modified LES, using a one-equation sub grid-scale model and the Sauer and Schnerr cavitation model, for scaled-down Francis turbine (hydrofoil). Cheng et al. (2021) conducted a simplified case study involving an elliptical NACA-16020 hydrofoil and introduced a novel cavitation model. They compared the numerical results with experimental observations and found that the gas content significantly influences cavitation phenomena. Miki et al. (2021) investigated cavitation formation and pressure distribution in non-cavitating and cavitating flow conditions for Delft twisted hydrofoil. In their work, a good correlation with experimental data is achieved for both the total cavity volume and the Strouhal number of cavitation shedding frequency, but the estimation of lift force fell short when compared to the experimental data. Yin et al. (2021) numerically investigated the unsteady cavitating flow around a symmetrical twisted hydrofoil. Their analysis focused on the cavitating flow characteristics, combining temporal and spatial aspects. In a research investigation conducted by Yin et al. (2022), they utilized the RANS method in combination with the k -epsilon turbulence model to examine how variations in entry angles and different cavitation numbers influence the lift and friction coefficients of a wing featuring a NACA4412 cross-section.

In the content of hydrofoil cavitation prediction, recent studies have employed various sophisticated computational methods and models to simulate flow patterns and cavitation phenomena, comparing them with experimental data for validation. Xie et al. (2023) investigated NACA0009 hydrofoil cavitation using Large Eddy Simulation (LES), Scale Adaptive Simulation (SAS), and Grid-Adaptive Simulation (GAS), noting SAS's inability to capture periodic cavitation behaviour. He et al. (2023) utilized LES with the Schnerr-Sauer cavitation model to predict cavitation over the Delft 11 hydrofoil, demonstrating good agreement with experimental observations. Ghasemnezhad and Roohi (2024) compared LES and k -omega turbulence models for NACA66 hydrofoil cavitation, considering pitching motions and various scenarios, while Liu et al. (2023) investigated S-shaped hydrofoil cavitation, validating simulations with experimental data using the FBDCM model. Pendar et al. (2023) focused on the twisting effect, employing LES to study flow around a sinusoidal leading edge NACA 634-021 foil, comparing results with a straight leading edge. Zhang et al. (2023) simulated NACA66 hydrofoil cavitation, comparing shedding structures using RANS, DES, and LES methods, noting RANS insufficiency for secondary shedding. Zhengdong Wang et al. (2023) analyzed cavitating flow around Delft hydrofoil using LES, Schnerr-Sauer cavitation, and k -omega turbulence models, comparing vapour volume with experimental data under different mesh configurations. Additionally, Ziyang Wang et al. (2023) explored cloud cavitation over the Delft hydrofoil, employing various cavitation bubble models via Lagrangian methods and comparing cavity behaviour with experimental results across different mesh setups. Furthermore, in the study where preliminary findings of this research are presented, Oksuz et al. (2023b) have conducted a numerical study on the investigation of the twisted profile effect on hydrofoils in cavitating flow conditions by RANS method.

The study aims to examine cavitation dynamics through the implementation of RANS and DES methods in the analysis of cavitating flows

around twisted hydrofoils. It seeks to evaluate the influence of the hydrofoil's curvature, specifically focusing on the effects of twist, on cavitation formation and hydrofoil performance through sophisticated numerical analyses. To accomplish this, a validation study for the Delft-11 hydrofoil is conducted initially. Subsequently, cavitating flow analyses are performed for three types of 3D hydrofoils: no-twist, half-twisted, and twisted, all featuring NACA0009 and NACA0015 foil sections. The study meticulously investigates cavitation development and drag and lift production on the hydrofoils across a range of angles of attack.

Following this introduction, the paper is structured into four additional sections as follows: Section 2 provides a detailed explanation of the CFD methodology conducted using both RANS and DES methods. Section 3 outlines the validation studies of non-cavitating and cavitating flow simulations of the Delft Twisted 11 hydrofoil. Chapter 4 encompasses the cavitating flow analysis of twisted, half-twisted, and no-twist hydrofoils, presenting results and discussions. Finally, Section 5 comprises the conclusions derived from the study.

2. Numerical setup

CFD analysis was conducted using the commercial software, Star-CCM+, to closely replicate the flow properties presented in the experiments. In order to simulate the flow characteristics around the hydrofoil, at the beginning, the incompressible and unsteady RANS (URANS) equations were initially discretized using the finite-volume method and were solved using a segregated approach. Subsequently, due to the desire for a more detailed modelling of cavitating flow dynamics and the inability to accurately obtain cavitation cycles using the RANS method, the study proceeded with the DES method.

2.1. Governing equations and turbulence modelling

The mass and momentum conservation equations are represented using Cartesian tensor notation as shown below (Ferziger et al., 2020):

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\tau_{ij} - \overline{\rho u_j u_i}) \quad (2)$$

In these equations, ρ , u , p and u' used as fluid density, averaged velocity, averaged static pressure and fluctuating velocity, respectively. τ_{ij} term, in Equation (2), represents the viscous component of the stress tensor while the term $\overline{\rho u_j u_i}$ describes the stresses generated by turbulent fluctuations, specifically known as Reynolds stresses. The overbar is used as a shorthand notation for averaging.

In our research, we model Reynolds stresses using the linear eddy-viscosity model. The linear constitutive relationship for this is presented in Equation (3) as follows.

$$-\overline{\rho u_j u_i} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij} \quad (3)$$

$$S_{ij} = \frac{1}{2}\rho \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (4)$$

In these equations, μ_t represents turbulent viscosity and k represents turbulent kinetic energy. δ_{ij} indicating the Kronecker delta and S_{ij} is used to represent mean strain rate tensor.

2.2. RANS and DES: Theoretical overview and comparison

RANS methods are known for their efficiency in simulating turbulent flows by averaging the flow variables over time. In the RANS method, the Navier-Stokes equations are solved in a time-averaged manner,

while the turbulence effects of all scales are modelled. While RANS models provide reasonably accurate predictions for many engineering applications, they are less suitable for capturing unsteady flow phenomena and resolving turbulent structures, particularly in regions of separated or highly sheared flows. This limitation can lead to deficiency predicting flow features such as vortex shedding, flow separation, or multiphase flow dynamics.

On the other hand, DES methods aim to overcome the limitations of RANS by combining elements of both RANS and LES approaches. DES employs a RANS model in regions of attached flow (boundary layer) and switches to a LES mode in regions of flow separation or unsteadiness, offering improved accuracy in resolving turbulent flow structures. In LES method, only the large-scale motions (large eddies) are resolved, while the small-scale motions are modelled. In LES, the equations are filtered (volume-averaged). The equations solved in LES are the same as those in RANS, but due to filtering in the solution, in other words, since large-scale turbulent terms are also resolved, the solution is more complex and challenging. Therefore, DES simulations can be computationally expensive, particularly for complex geometries and high Reynolds number flows, which may limit their practicality for certain engineering applications.

2.2.1. Mathematical formulation of RANS

RANS is based on time-averaging the Navier-Stokes equations to model turbulent flows. The fundamental approach is to decompose the instantaneous velocity field u into a mean component $\bar{u}$ and a fluctuating component u' :

$$u = \bar{u} + u' \quad (5)$$

The time-averaged Navier-Stokes equations are written as follows:

$$\bar{u} \cdot \nabla \bar{u} = -\frac{1}{\rho} \nabla \bar{p} + \mu \nabla^2 \bar{u} + \text{RANS terms} \quad (6)$$

The turbulence effects are modelled through the Reynolds stress tensor $\overline{u' u'}$, which is approximated using turbulence models such as the $k-\epsilon$ or $k-\omega$ models:

$$\overline{u' u'} = \mu_t (\nabla \bar{u} + \nabla \bar{u}^T) - \frac{2}{3} k \mathbf{I} \quad (7)$$

Here, $\mathbf{I}$ denotes the identity matrix.

2.2.2. Mathematical formulation of DES

DES integrates LES with RANS, using RANS near walls and LES in the free stream to combine the advantages of both methods. The governing equations for DES are similar to those of LES but with a modified turbulence model:

$$\frac{\partial u}{\partial t} + u \cdot \nabla u = -\frac{1}{\rho} \nabla p + \mu \nabla^2 u + \mathbf{S} \quad (8)$$

where $\mathbf{S}$ represents the subgrid-scale (SGS) stress tensor, which is modelled differently based on whether the flow is in the RANS or LES region. DES uses:

$$\mathbf{S} = \tau_{SGS} - \frac{2}{3} \text{tr}(\tau_{SGS}) \mathbf{I} \quad (9)$$

where τ_{SGS} represents the SGS stress tensor. The transition between RANS and LES regions is governed by a blending function, usually a function of the local mesh size and turbulence length scales.

For more detailed information, please refer to Wilcox (2006).

2.2.2.1. Resolution of turbulent scales. DES provides a more detailed resolution of turbulent structures compared to RANS. In RANS, the turbulence modelling approximates the Reynolds stresses using closure models, which can introduce significant errors, particularly in complex

flows where unsteady or large-scale structures are prominent. Mathematically, RANS models are limited by the accuracy of the turbulence model used to approximate $\overline{u' u'}$.

2.2.2.2. Local accuracy vs. global efficiency. DES uses LES to capture large, energy containing eddies while relying on RANS near walls to efficiently model boundary layers. This approach allows DES to better resolve flow features such as separation and recirculation zones, where RANS often fails to provide accurate predictions. Mathematically, DES's hybrid approach improves accuracy without excessively increasing computational cost, as the LES region dynamically adapts to flow features.

2.2.2.3. Turbulent mixing and transition. DES effectively models turbulent mixing and transition between different flow regimes. The mathematical formulation of DES's blending function ensures that it switches between RANS and LES based on local flow characteristics, capturing complex interactions that RANS models might miss. The blending function β is typically defined as:

$$\beta = \frac{1}{1 + \left(\frac{L_{DES}}{L_{LES}} \right)^f} \quad (10)$$

Where, L_{DES} and L_{LES} are the turbulence length scales in the RANS and LES regions, respectively, and f is a parameter that controls the transition.

In summary, DES offers a mathematically superior approach to modelling turbulent flows compared to RANS. By combining LES and RANS, DES leverages the strengths of both methods, providing improved resolution of turbulent structures and better accuracy in capturing complex flow phenomena. The mathematical formulations of DES allow for more accurate representation of turbulent stresses and transitional flows, addressing many of the limitations inherent in traditional RANS models.

In this paper, CFD analyses were conducted using unsteady RANS and DES techniques, combined with the *Shear-Stress Transport (SST) k-Omega turbulence model* for both. In the simulations, the preference was given to the *Two-Equation SST k- ω turbulence model*, which utilizes the *Boussinesq approach* to represent Reynolds stress tensors. In this particular model, the turbulence viscosity is determined by solving equations that govern turbulent kinetic energy (k) and its rate of energy dissipation.

The *K-Omega turbulence model* is a two-equation model that solves transport equations for the turbulent kinetic energy (k) and the specific dissipation rate (ω).

Wilcox (2006) served as the most extensive resource on the *K-Omega model*, covering its inception, comparing it with alternative models, and introducing the most recent iteration of the model. Being the creator of the *K-Omega model*, Wilcox asserts the superiority of his model over the *K-Epsilon model* and advocates for the superiority of the *Omega transport equation* over other scale equations. An acknowledged benefit of the *K-Omega model* compared to the *K-Epsilon model* is its enhanced capability in handling boundary layers experiencing adverse pressure gradients. However, its most notable advantage lies in its applicability across the entirety of the boundary layer, encompassing the viscous-dominated region, without requiring additional adjustments (Star CCM, 2022).

The turbulent eddy viscosity (μ_e) is calculated as:

$$\mu_e = \rho k T \quad (11)$$

Where, ρ is the fluid density and T is the turbulent time scale.

Transport equations for the *K-Omega turbulence model* is presented as follows:

$$\frac{\partial}{\partial t}(\rho k) + \nabla \cdot (\rho k \bar{v}) = \nabla \cdot [(\mu + \sigma_k u_t) \nabla k] + P_k - \rho \beta^* f_{\beta^*} (\omega k - \omega_0 k_0) + S_k \quad (12)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \nabla \cdot (\rho \omega \bar{v}) = \nabla \cdot [(\mu + \sigma_\omega u_t) \nabla \omega] + P_\omega - \rho \beta f_\beta (\omega^2 - \omega_0^2) + S_\omega \quad (13)$$

In the equations provided, $\bar{v}$ represents the mean velocity, μ stands for dynamic viscosity, σ_k and σ_ω denote model coefficients, P_k and P_ω represent production terms, f_{β^*} signifies the free-shear modification factor, β^* denotes the *k-Omega model* coefficients, f_β indicates the vortex-stretching modification factor, S_k and S_ω are the user-specified source terms and finally, k_0 and ω_0 are the ambient turbulence values that counteract turbulence decay. More information regarding the *K-Omega turbulence model* can be found in the (Star CCM, 2022).

2.2.3. SST (menter) K-Omega Detached Eddy model

In this study, the solution of the Reynolds stress tensor around the boundary layer was conducted using the *SST K-Omega Detached Eddy model* together with the *All y + Wall treatment* wall function for the near-wall region (Star CCM, 2022). *SST K-Omega Detached Eddy model* combines features of the *SST K-Omega RANS model* in the boundary layers with a LES in unsteady separated regions.

The DES version of the *SST K-Omega model* is achieved through alterations made to the dissipation term within the transport equation governing turbulent kinetic energy (Star CCM, 2022). In the outer region, the sub grid stresses were resolved using the LES Smagorinsky model. The turbulent length scales are determined using the IDDES turbulent length scale method.

2.2.4. IDDES formulation

For the IDDES formulation of Shur et al. (2008), the specific dissipation rate ω in Eq. (12) is replaced by $\tilde{\omega}$ defined as:

$$\tilde{\omega} = \frac{\sqrt{k}}{l_{HYBRID} f_{\beta^*} \beta^*} \quad (14)$$

$$l_{HYBRID} = \tilde{f}_d (1 + f_e) + l_{RANS} + (1 - \tilde{f}_d) C_{DES} \Delta_{IDDES} \quad (15)$$

$$l_{RANS} = \frac{\sqrt{k}}{f_{\beta^*} \beta^* \omega} \quad (16)$$

$$l_{IDDES} = \tilde{f}_d (1 + f_e) l_{RANS} + (1 - \tilde{f}_d) l_{LES} \quad (17)$$

$$l_{LES} = C_{DES} \Delta \quad (18)$$

$$l_{RANS} = \frac{\sqrt{k}}{C_\mu \omega} \quad (19)$$

$$C_{DES} = C_{DES1} F_1 + C_{DES2} (1 - F_1) \quad (20)$$

In these formulations, k represents the turbulent kinetic energy while Δ denotes the largest distance between the cell centre under consideration and the cell centers of the neighboring cells. More information about the mathematical background of IDDES model can be found in Gritskevich et al. (2012) and (Star CCM, 2022).

2.3. Cavitation modelling

In the current modelling approach chosen in the flow solver, the cavitating flow is primarily focused on two substances: water and vapour. It does not take into account the presence of non-condensable gases. To address this, two distinct methods are suggested to address the equations governing these two substances, referred to as the "inhomogeneous" and "homogeneous" methodologies (Yin et al., 2021).

To model the two-phase cavitating flow, the method involving a homogeneous mixture was employed. This approach focuses on the existence of seed particles and their subsequent expansion and contraction, forming bubbles. It assumes that both phases are thoroughly mixed, moving at identical speeds, experiencing the same pressure, and maintaining local thermodynamic equilibrium by having the same temperature (Usta and Korkut, 2019). Therefore, water and vapour are treated as a single fluid mixture.

The cavitation formation was simulated using the Schnerr-Sauer cavitation model. In accordance with this model, bubbles are treated as spherical entities, and all initial seeds possess identical radii at the start of the simulations.

The Schnerr-Sauer model is based on the assumption that the bubbles have a spherical shape, their growth is primarily governed by inertial forces and all initial seeds possess identical radii at the start of the simulations. The mathematical equation for the bubble radius (R_{bubble}) in the Schnerr-Sauer cavitation model is as follows:

$$R_{bubble} = \left(\frac{\alpha_v}{1 - \alpha_v} \cdot \frac{3}{4\pi} \cdot \frac{1}{n_{bubble}} \right)^{1/3} \quad (21)$$

In Eq. (21), α_v is vapour volume fraction and n_{bubble} is the number of bubbles per unit volume of liquid. For further information about the Schnerr-Sauer cavitation model, please refer Usta and Korkut (2019).

2.4. Lift, drag and pressure coefficients

The foil efficiency is defined by the lift and drag coefficients which are dimensionless quantities. The lift coefficient describes the lift force produced by foil divided by the flow velocity, section area, and density. Similarly, the drag coefficient means the drag force applied to the foil by fluid divided by flow velocity, planform area, and density. These coefficients are used by researchers for design optimization, performance analysis, and hydrodynamic analyses. Lift and drag coefficients of the foils are predicted by using experimental methods (in a water tunnel) or simulations. These values change depending by various angle of attacks and flow conditions. The lift coefficient, drag coefficient, pressure coefficient and Reynolds number formulas are given below.

$$C_D = \frac{D}{0.5\rho A(U)^2} \quad (22)$$

$$C_L = \frac{L}{0.5\rho A(U)^2} \quad (23)$$

$$C_p = \frac{P - P_\infty}{0.5\rho(U)^2} \quad (24)$$

$$Re = \frac{u \cdot c}{\nu} \quad (25)$$

In these equations, C_D represents the drag coefficient, D denotes the drag force in Newtons (N); ρ , U and A represent the density of the fluid ($\frac{kg}{m^3}$), velocity of the flow ($\frac{m}{s}$) and planform area in square meters (m^2), respectively. C_L is the lift coefficient, while L represents the lift force (Newton). P is reference pressure (Pa), P_∞ is the liquid pressure at infinity and A is the projected surface area (m^2).

3. Validation studies: Non-cavitating and cavitating flow simulations of Delft Twisted 11 hydrofoil

The benchmark experimental study, which is validated in this paper is the research of Foeth (2008a). In this study, cavitation tunnel tests were carried out on a three-dimensional twisted hydrofoil, known as the Delft Twisted 11 Hydrofoil (NACA0009 twisted at -2° angle of attack), at the University of Delft (Foeth, 2008a). Comprehensive measurements were carried out in this study, to assess the forces and moments exerted by the flow on the hydrofoil, which is enclosed at the tips by the walls of

the cavitation tunnel. Additionally, local pressure data on the hydrofoil's surface were also recorded. Validation has been accomplished for both non-cavitating flow conditions, referred to as "Case 1," and cavitating flow conditions, referred to as "Case 2," in this study.

3.1. Geometry, flow conditions and boundary conditions

The Delft Twisted 11 Hydrofoil features a rectangular wing with a uniform cross-sectional shape spanning its entire length, while its orientation with respect to the incoming flow varies gradually along its span. The angle of attack is defined with respect to the tip sections of the wing. When the angle of attack is 0° at both ends of the foil, it is 11° at the mid-section. The spanwise angle-of-attack variation of the section for Delft Twisted 11 Hydrofoil given by (Hoekstra et al., 2011);

$$\alpha(y) = \alpha_{max} \left[2(|y - 1|)^3 - 3(|y - 1|)^2 + 1 \right] \quad (26)$$

Where; $\alpha_{max} = 11^\circ$, y is the coordinate in the direction of the span, ranging from 0 to 2.

Geometry of the hydrofoil is presented in Fig. 1. The chord length of the hydrofoil is $c = 0.15$ m, while span length is $b = 0.3$ m.

The size of computational domain was generated utilizing some studies in the literature that validates the benchmark Delft Twist 11 Hydrofoil experiments as well (Koop, 2008; Hoekstra et al., 2011; Melissaris et al., 2019; Usta and Korkut, 2019; Oksuz et al., 2023a). Specifically, the distance from the domain's inlet to the hydrofoil's leading edge was set at 2 times the hydrofoil's chord length (see Fig. 2, denoted as "c"). Meanwhile, the distance between the hydrofoil's trailing edge and the domain's outlet was maintained at 4 times the chord length ("4c"). Additionally, the hydrofoil was positioned at a distance of one chord length from both the bottom and top walls of the domain.

The same computational domain was used for non-cavitating (Case 1) and cavitating (Case 2) flow conditions. The physical conditions were replicated as in the numerical studies performed by Hoekstra et al. (2011), Koop (2008) and Oksuz et al. (2023a).

In this study, Case 2 simulations have been repeated using the DES method. The details of the DES analyses will be provided in the subsequent sections.

Flow conditions for Case 1 and Case 2 simulations are tabulated in Table 1.

In all numerical flow problems, it's essential to establish initial and boundary conditions that align with the specific characteristics of the problem being addressed. The flow was introduced through a velocity inlet, and its end was designated as the pressure outlet boundary condition. The top, bottom and side wall of the domain were treated as slip walls, while hydrofoil was set to no-slip wall condition. To simplify the computational process and reduce solution time, only half of the hydrofoil was actually modelled. This is achieved by employing symmetry plane boundary condition, which accurately replicates the behaviour of the other half of the computational domain. Fig. 2 provides a three-

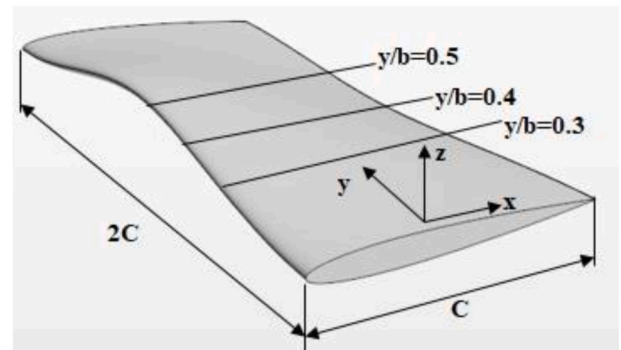


Fig. 1. Geometry of the delft twist 11 hydrofoil.

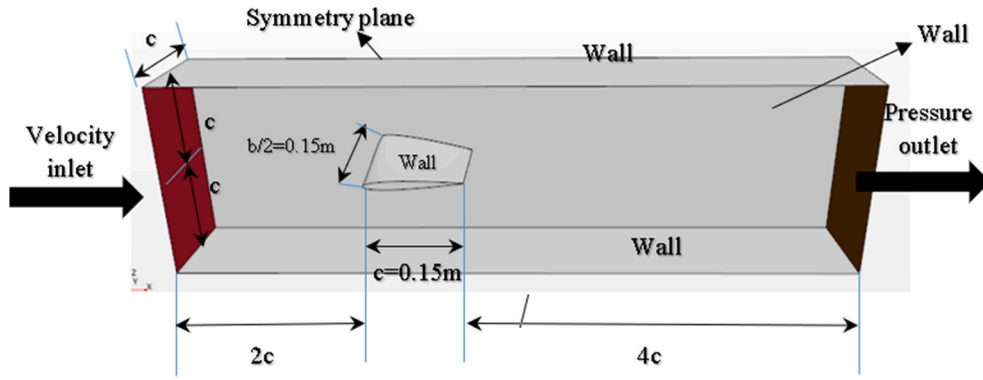


Fig. 2. Size of the computational domain and boundary conditions.

Table 1

Flow conditions in the Delft twisted hydrofoil simulations.

	Angle of attack (deg)	Outlet pressure (kPa)	Vapour density (kg/m ³)	Inlet velocity (m/s)	Reynolds Number	Vapour pressure (kPa)	Water density (kg/m ³)	Cavitation number
Case 1	-2	97	–	6.75	1.14×10^6	–	998	–
Case 2	-2	29	0.023	6.97	1.18×10^6	2.970	998	1.07

dimensional depiction for the computational domain and boundary conditions.

3.2. Grid generation by considering the CFL number

The mesh on the hydrofoil and time step value were determined by corresponding the CFL stability criterion. This criterion is based on the Courant number (CFL), which represents the ratio between the physical time step and the mesh convection time scale. It establishes a connection between the size of the mesh cells (Δx), time step (Δt) and the velocity of

flow within the mesh. This relationship is expressed as follows:

$$\text{Courant number} = \left| \frac{U \Delta t}{\Delta x} \right| \quad (27)$$

The Courant number was computed for each simulation, and this calculation was performed to satisfy the following CFL criteria.

$$\text{CFL criteria} = \left| \frac{U \Delta t}{\Delta x} \right| \leq 1 \quad (28)$$

The computational domain was discretized using an unstructured

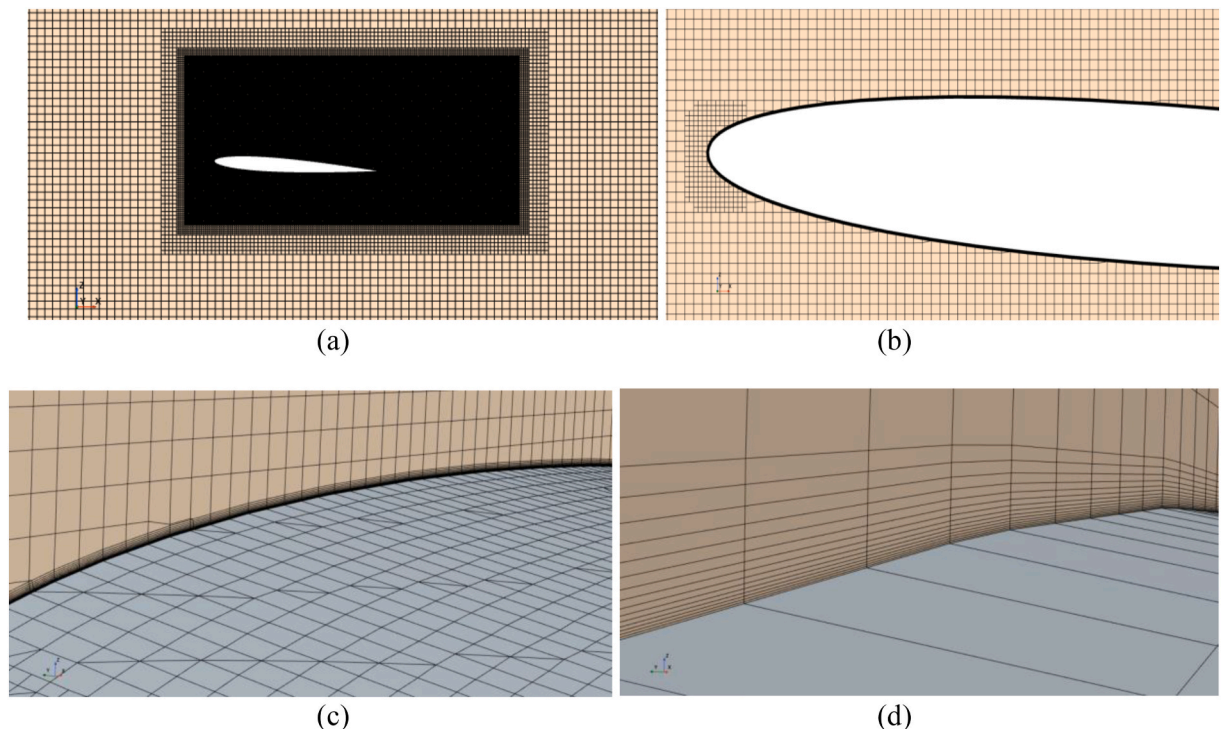


Fig. 3. Grid configuration of the (a) computational domain, (b) hydrofoil surface, (c) hydrofoil surface and boundary layer, (d) boundary layer prism mesh.

grid composed of hexahedral elements. To enhance solution accuracy, efforts were made to minimize cell skewness and achieve smooth mesh transitions, thereby reducing numerical errors in the simulations.

For RANS analyses, the computational grid used in the simulation had around 4.5 million cells which is determined with grid independence and grid convergence studies utilizing the GCI method given in Oksuz et al. (2023a).

In DES analyses, the same mesh models as those in RANS analyses were employed. The only difference in the analyses lies in the mesh density. Particularly in the regions away from the hydrofoil, DES analyses have better modelled cavitation dynamics compared to RANS. Around the boundary layer, the higher mesh density resulted in DES yielding more accurate representation of the flow. Ultimately, in the Unsteady DES flow simulations, approximately 12 million total grid elements were employed within the computational domain.

Fig. 3 displays the configuration of grid, where cell sizes are smaller in the vicinity of the hydrofoil. This finer mesh near the hydrofoil was employed because the velocity shows significant changes in this area. Conversely, as one moves farther away from the hydrofoil, the cell sizes increase. The same grid was used in Case 1 and Case 2 flow conditions in RANS analysis.

Grid refinement levels on the computational domain are seen in Fig. 3.

As seen from Fig. 3, uniform grid cells of the same size are generated in the regions away from the geometry within the computational domain. Mesh refinement has been conducted as the hydrofoil is approached, resulting in densification of the mesh. A highly refined mesh is established on the hydrofoil surface to accurately capture cavitation phenomena. Prism layers are constructed to accurately model the flow dynamics within the boundary layer mesh, ensuring that the y^+ value remains below 1.

As cavitation did not occur under the flow conditions of Case 1, the analyses were conducted under steady conditions by using RANS method. For more comprehensive information regarding flow conditions in Case 1, please refer to Oksuz et al. (2023a).

The way we divide time into discrete intervals directly impacts the level of error introduced in the simulation, and it is a critical factor in achieving accurate time-dependent computations. The time step is chosen as 1×10^{-5} s for all unsteady flow scenarios, where flow properties change over time, and it sets the duration of each iteration in the solution process. Moreover, the time discretization is selected as second-

order since high-accuracy discretization schemes facilitate quicker unsteady solutions by enabling the use of larger time steps while maintaining precision (Star CCM, 2022).

Since modelling the cavitation flow is challenging, special attention has been given to ensuring the satisfaction of the Courant condition in Case 2. The distribution of the Courant number over the hydrofoil for Case 2 shown in Fig. 4.

As indicated by Fig. 4, it is apparent that both RANS and DES analyses fulfil the Courant number condition quite substantially. Due to the utilization of denser mesh in DES analyses, the Courant number is smaller compared to RANS analyses.

In the DES method, the resolution of where the model performs RANS or LES calculations is determined by the density of the solution mesh. Therefore, it is essential to generate the mesh in accordance with the DES model to obtain good results from the DES model. If the maximum size of the mesh is larger than necessary, there will be no transition from RANS to LES in the model, meaning that the required and sufficient scales cannot be resolved. In such cases, there would be no advantage to using the DES model, and the analysis conducted would not be a DES solution (Usta, 2018).

3.3. Validation of the non-cavitating flow simulations (case 1)

The validation for Case 1 study was conducted by comparing the obtained results with the experimental data published by Foeth (2008b). Foeth's study did not provide the drag coefficient value due to its notably high uncertainty. Therefore, the drag coefficient has been compared to numerical studies available in the literature. The lift coefficient value of Foeth (2008b) is calculated as 0.444, for the $L = 455$ N at 6.75 m/s inflow velocity, by using Eq. (23). Comparison of the lift and drag coefficients are presented in Table 2.

Table 2

Comparison of the drag and lift coefficients obtained in the present study with those reported in the literature (Case 1).

	Foeth (2008b) (Experiment)	Koop (2008) (CFD)	Vaz et al. (2017) (CFD)	Asnaghi et al. (2018) (CFD)	Present study
C_D	–	0.0083	0.0143–0.0146	0.0120–0.0140	0.0130
C_L	0.444	0.445	0.425–0.428	0.453–0.455	0.4364

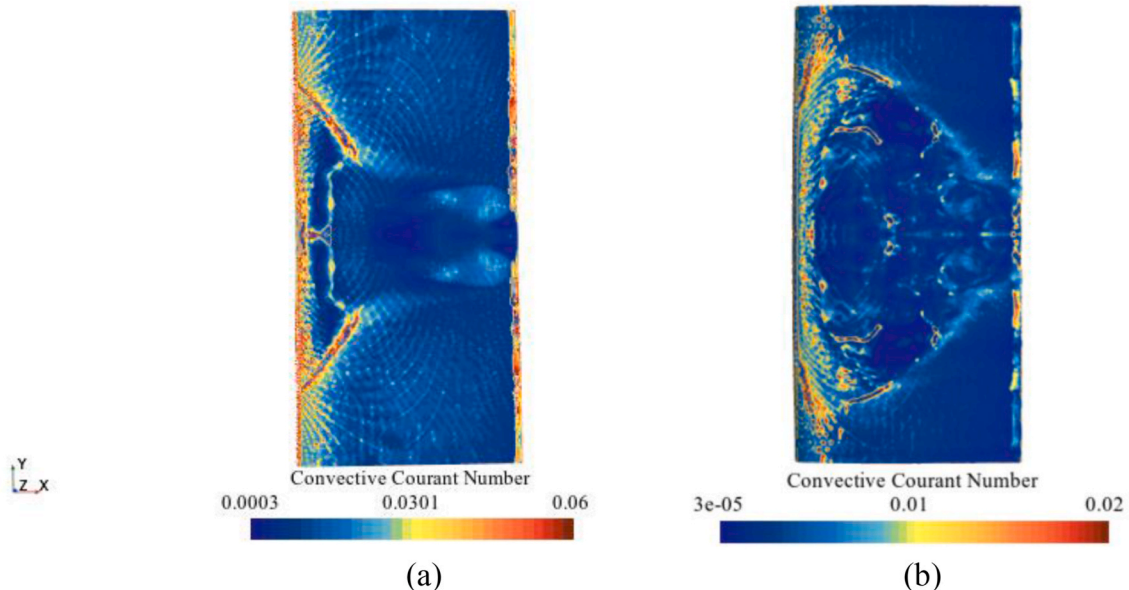


Fig. 4. Courant number distribution over the hydrofoil (a) RANS simulations (mesh with 4.5 million cells), (b) DES simulations (mesh with 12 million cells).

According to the values presented in Table 2, the obtained drag coefficient values are consistent with those from other CFD studies in the literature. Additionally, the obtained drag coefficient values closely align with CFD results from other studies in the literature.

Comparison of pressure coefficient distributions with the experimental data of Delft Twist 11 Hydrofoil (Foeth, 2008b) at mid span ($y/b = 0.5$), $y/b = 0.4$ and $y/b = 0.3$ are presented in Fig. 5. The parameter b represents the full span width. Therefore, y/b denotes the mid span width.

It is seen from Table 2 and Fig. 5 that the drag, lift and pressure coefficients produced by the hydrofoil in this study are in good

agreement with experimental and numerical results reported in the literature.

3.4. Validation of the cavitating flow simulations (case 2)

The results obtained under the cavitation conditions of Case 2 for the validation of the Delft Twisted 11 Hydrofoil are presented in the table below, alongside experimental data and numerical findings from existing literature (Table 3). Cavitating flows were solved using Unsteady Reynolds-Averaged Navier-Stokes (URANS) and Detached Eddy Simulation (DES) methods. The turbulence and momentum equations were discretized using the first-order upwind scheme for URANS, while the second order upwind scheme is used for DES. The time step is chosen as $\Delta t = 1 \times 10^{-5}$ s at all of the unsteady flow simulations performed for validation.

According to the literature, (Foeth, 2008b), the drag force was not measured in the experiments conducted for Case 2. Upon examining the drag values provided in Table 3 and it is observed that the result obtained from RANS analysis in this study is consistent with the findings in the literature, whereas the result from DES analysis yields a drag force higher than the literature values. It is believed that this discrepancy may be attributed to the disruption of flow caused by the cavitation structure formed on the hydrofoil. When comparing the lift coefficient values obtained in the study, it is observed that the lift coefficient values obtained from both RANS and DES analyses are in line with the literature, in addition, the value obtained from the DES analysis is the closest to the experimental result.

Cavitation development on the hydrofoil is compared with those of experiment in Fig. 6. Volume fraction ratio is taken as 0.5 generating the cavitation volume scenes.

Although the general trend of cavitation formation in the RANS method is similar to the experimental results, it shows less cavitation development. However, as clearly seen in Fig. 6, the cavitation pattern obtained with DES closely resembles the visual obtained from the experiment.

Comparing the lift and drag coefficients (Table 3) and the cavitation development on the hydrofoil, (Fig. 6), it can be said that the results are similar to the results given the literature. The RANS method does not sufficiently capture the cloud cavitation behind the sheet cavitation. Based on this, it can be concluded that the influence of cloud cavitation on the lift of the examined hydrofoil is assessed to be minimal.

The results from analyses by using RANS with 4.5 million mesh cells show a good agreement in terms of the drag and lift coefficients in the literature. As shown in the next section, while the cavitation volume converges to an approximately constant value after a certain time in URANS solutions, the cycle and period of cavitation occurrence on the hydrofoil surface can be clearly seen in DES solutions (Figs. 10 and 11).

After the analysis results are obtained by discretizing using the first-order upwind scheme for URANS, the second-order upwind scheme is applied for the same mesh configuration, the cavitation volume and force coefficients are found approximately the same in both schemes. Thus, it is concluded that the first-order upwind scheme for URANS and the number of 4.5 million mesh elements are sufficient for the estimation of lift force and average cavitation volume image in time-dependent cavitation analysis. However, the determination of the coherent period, which represents the natural structure of cavitation, could not be

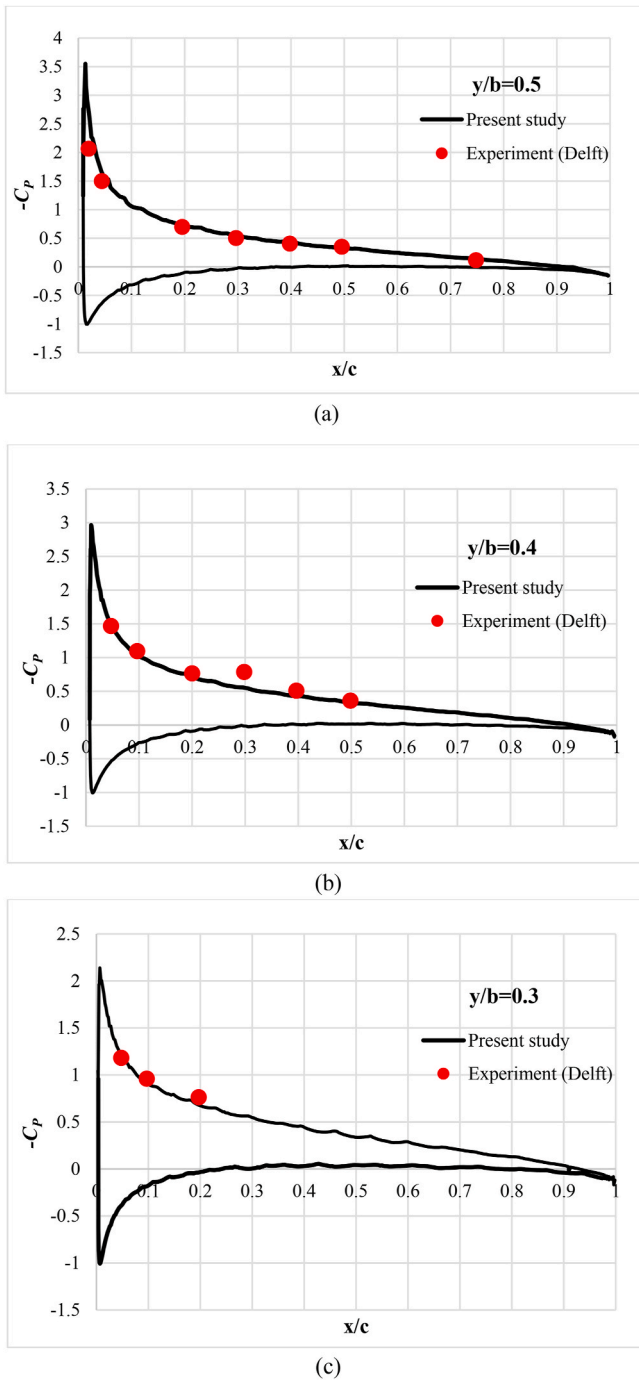


Fig. 5. Comparison of pressure coefficient distributions with the experimental data of Delft Twist 11 Hydrofoil (Foeth, 2008b) at $y/b = 0.5$ (mid span) (a), $y/b = 0.4$ (b) and $y/b = 0.3$ (c) for non-cavitating flow.

Table 3

Comparison of the drag and lift coefficients obtained in the present study with those reported in the literature (Case 2).

	Foeth (2008b) (Experiment)	(Whitworth, 2011) RANS	(Usta, 2018) DES	Present study RANS	Present study DES
C_D	–	0.0242	0.0266	0.0236	0.0401
C_L	0.523	0.403	0.433	0.428	0.443

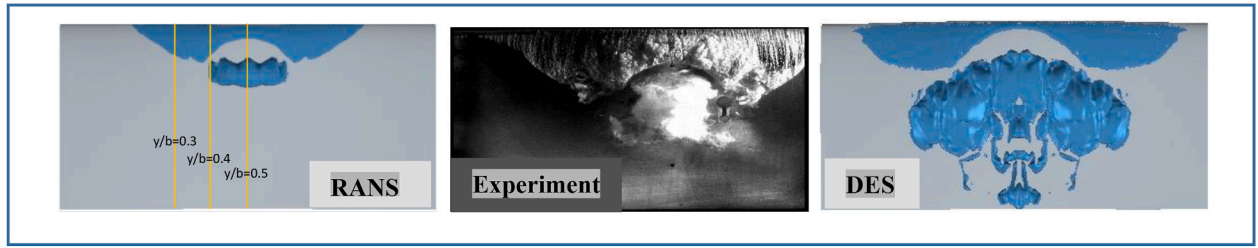


Fig. 6. Cavitation development obtained in present study (left and right) and experiment (Foeth, 2008a) (middle).

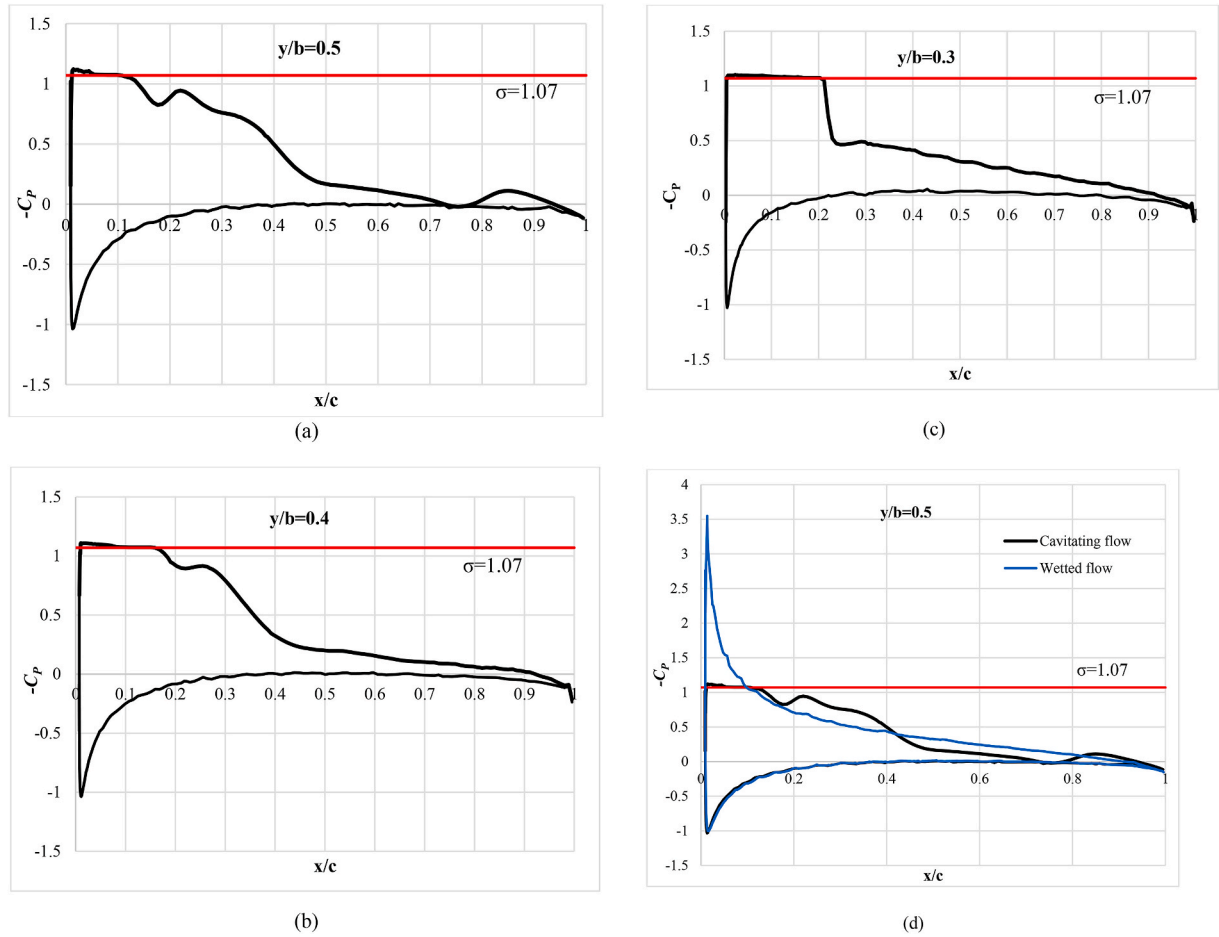


Fig. 7. Pressure coefficient distributions of NACA0009 twisted hydrofoil at $\text{AoA} = -2^\circ$ at $y/b = 0.5$ (mid span) (a), $y/b = 0.4$ (b), $y/b = 0.3$ (c) for cavitating flow and $y/b = 0.5$ (mid span) (d) for non-cavitating and cavitating flow.

achieved using RANS. For this reason, the number of elements was increased to 12 million and RANS simulations were performed to obtain the cavitation period accurately. However, cavitation period could not be obtained even with RANS analyses consisting of a 12 million element mesh. Therefore, the remaining part of the study utilized the DES method. For this reason, in the section below, DES method was used instead of RANS and the analyses were continued to determine the cycle and period of cavitation occurrence using the second-order upwind scheme with the same refined mesh structure. For cavitating flow analyses by RANS, pressure coefficient distributions NACA0009 twisted hydrofoil ($\text{AoA} = -2^\circ$) at mid span ($y/b = 0.5$), $y/b = 0.4$ and $y/b = 0.3$ (please see Fig. 6) are presented in Fig. 7.

From the pressure distribution graphs at different sections of the hydrofoil shown in Fig. 7 (a), (b), (c), it can be seen that the cavitation regions are consistent with the cavitation number (σ) and the cavitation volume image obtained from RANS in Fig. 6. As can be seen from the

pressure distribution graphs on the hydrofoil mid-section with cavitating and non cavitating flow conditions, the pressure distributions on the pressure side does not change due to cavitation. As expected, the negative pressure coefficient values near the tip of suction side where cavitation occurs are equal to the cavitation number (σ).

4. Cavitating flow analysis of twisted, half-twisted and No-twist hydrofoils

4.1. Generated geometries

After achieving successful validations for the Delft hydrofoil (NACA0009 Twisted $\text{AoA} = -2^\circ$), this section focuses on generating twisted, half-twisted, and no-twist 3D hydrofoils for NACA0009 and NACA0015 hydrofoils. Subsequently, wetted (non-cavitating) and cavitating flow analyses were conducted for various angles of attack

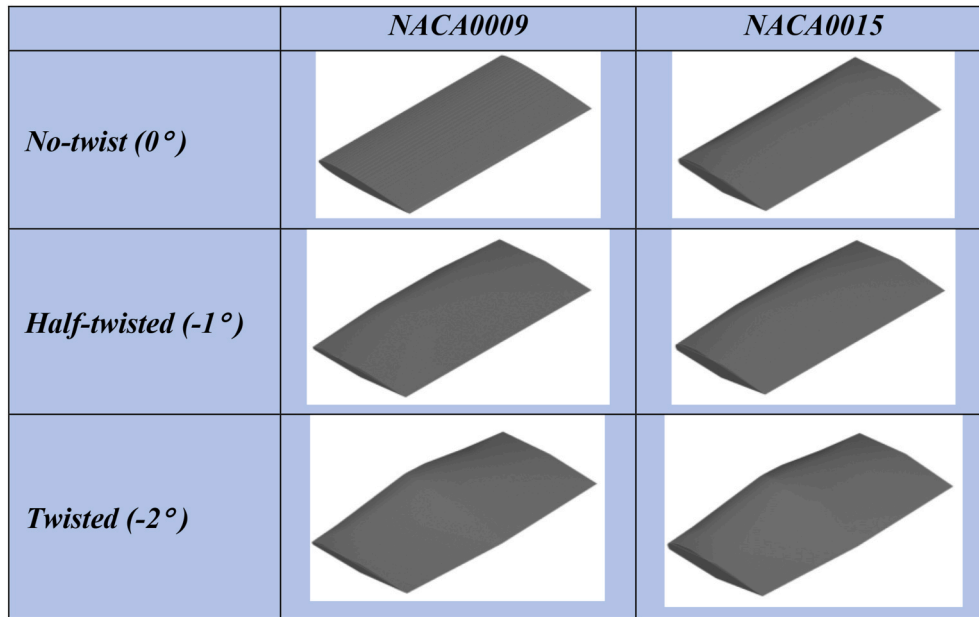


Fig. 8. NACA0009 and NACA0015 hydrofoil geometries used in the paper.

under the same flow conditions outlined in Table 1.

All the hydrofoil geometries used in the study have a span of 0.30 m and a chord length of 0.15 m, resulting in an aspect ratio of 2. Specifically, at the tip section of the hydrofoils, the geometric angles of attack are set to -2° for the twisted hydrofoils and -1° for the half-twisted ones, as illustrated in Fig. 8.

The hydrofoil geometries were generated considering the geometrical and mathematical information in Abbot et al. (1945). The half-twisted hydrofoil was created by taking half of the twisted hydrofoil, whereas the twisted hydrofoil's angle of attack distribution belongs to the Delft Twist-11 hydrofoil. The other geometrical details can be found in Oksuz et al. (2023a).

The geometry curvature given in the study varies between -2 (foil tip) and 9 (foil centre) degrees. Since the angle values given in the graphs as angle of attack are 11° at the foil centre, this value should be considered as $+ \text{AoA}$. For example, in the twisted geometry cases, the value in the centre of the hydrofoil at the angle of attack $\text{AoA} = +2^\circ$ is $2 + 11 = 13^\circ$. In the half-twisted geometry cases, since the angle values are halved, the angle value at the centre of the wing in the case of $\text{AoA} = +2^\circ$ is $\text{AoA} = 2 + 5.5 = 7.5^\circ$.

In the CFD analyses, the planform area A for all the twisted, half-twisted, and no-twist hydrofoils was taken to be $A = \text{chord} \times \text{span}$ is equal for all configurations.

4.2. The lift and drag coefficient results of twisted and half-twisted hydrofoils for various angle of attack conditions

Comparative analysis of the lift coefficients, drag coefficients and lift to drag ratio values of twisted and half-twisted geometries under non-cavitating and cavitating flow conditions are presented in Fig. 9. The experimental data for lift coefficient given by Foeth (2008b) is included to Fig. 9 (a) to validate the non-cavitating flow results of the twisted NACA0009 hydrofoil. The non-cavitating flow simulations were conducted under steady conditions, while the cavitating flow conditions of twisted hydrofoils were simulated using RANS with a time step of $\Delta t = 1 \times 10^{-5}$ s.

As seen in Fig. 9 (a), the twisted hydrofoil with NACA0009 section consistently generates higher lift up to approximately 2 degrees of angle of attack under the same cavitation conditions. In the case of cavitating flow, it is observed that the lift coefficient is approximately 25% lower

for the NACA0015 between -2 and $+2$ angles of attack, which is thicker twisted hydrofoil. The maximum lift coefficient values in non-cavitating flow occur at around 2° for NACA0009 twisted hydrofoil ($\text{AoA} = 13^\circ$ at the mid-section of the hydrofoil) and 3° for NACA0015 twisted hydrofoil ($\text{AoA} = 14^\circ$ at the mid-section of the hydrofoil). As can be seen from the lift coefficient graphs obtained for half-twisted hydrofoils (Fig. 9 (b)), in cavitation conditions, both hydrofoils produce less lift than the ones at wetted flow, with this effect being more pronounced in the thicker NACA0015 section hydrofoil. Similar to twisted hydrofoils, the half-twisted NACA0009 hydrofoil in non-cavitating conditions produces maximum lift at an earlier angle (approximately 5° , $\text{AoA} = 10.5^\circ$ at the mid-section of the hydrofoil). As seen in Fig. 9 (c), while the twisted NACA0009 hydrofoil generates slightly higher drag in cavitation conditions, in non-cavitating conditions, up to approximately 2 degrees of angle, the drag coefficients are very close, but beyond this, unlike the decrease in lift coefficient at higher angles, the C_D values for NACA0009 twisted hydrofoil dramatically increase. For half-twisted hydrofoils, the drag coefficients are higher for the NACA0015 in cavitation conditions, contrary to the trend in twisted hydrofoils. Similar to the twisted case, the half-twisted hydrofoil NACA0009 produces more drag after approximately 5 degrees of angle in non-cavitating condition (Fig. 9 (d)). In the case of non-cavitation flow, the thinner hydrofoil can give a higher drag coefficient at relatively high angles of attack (Seeni, 2022; Dongli et al., 2015; Sarraf et al., 2007; Sogukpinar, 2017; Xu and Wu, 2023).

As interpreted from Fig. 9 (e), which presents the C_L/C_D ratios of twisted hydrofoils, in non-cavitating conditions, the thinner section hydrofoil provides higher efficiency up to approximately 2° . Maximum efficiency values for both hydrofoils occur around stall angles. For the cavitating flow condition, the efficiency values approach each other as the angle of attack increases for twisted hydrofoils, while the efficiency of the thinner hydrofoil is significantly higher for half-twisted cases. In non-cavitating conditions, the efficiencies of half-twisted hydrofoils are approximately the same up to 3° , but after approximately 5° , the efficiency of the thinner hydrofoil dramatically decreases. Maximum efficiency values for both hydrofoils occur between 3 and 5 degrees of angle of attack (Fig. 9 (f)). In cavitating flow, the maximum hydrofoil efficiency in twisted hydrofoils occurs at -2° angle of attack, while in half-twisted hydrofoils it occurs at about 1° angle of attack. The efficiencies of cavitating and non-cavitating hydrofoils can be compared for a

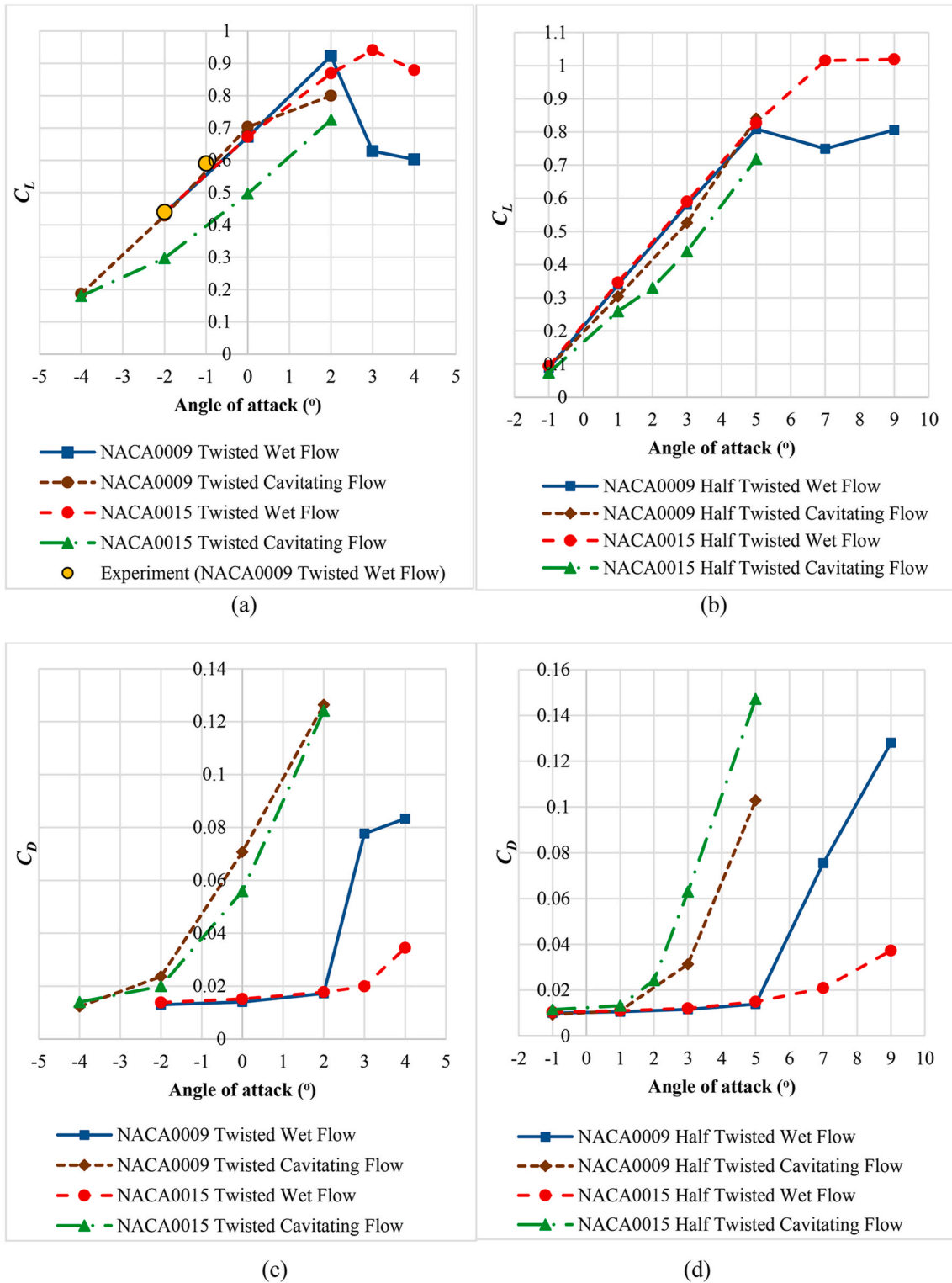


Fig. 9. (a,b) Lift coefficients, (c,d) Drag coefficients and (e,f) Lift to Drag ratio values of twisted and half-twisted geometries in non-cavitating and cavitating flow.

constant lift coefficient. As an example, in Fig. 9 (a), $C_L = 0.7$ is seen at wet flow around 0.2° angle of attack for both twisted NACA0009 and NACA0015. For cavitating flow, the angle of attack should be increased by 1.8° for NACA0015 to produce the same lift force as NACA0009. The lift coefficients for half-twisted geometries are presented in Fig. 9(b). Similarly, the $C_L = 0.7$ lift coefficient corresponds to the 3.9° angle of attack for both NACA0009 and NACA0015 at wet flow conditions. In

cavitating flow, the same lift coefficient is seen at 4.2° angle of attack for NACA0009 and while 4.9° angle of attack for the NACA0015 section.

The hydrofoil efficiencies corresponding to the angles of attack obtained for $C_L = 0.7$ from Fig. 9 (a) and 9 (b) are read from graphs in Fig. 9 (e) and 9 (f). Namely, hydrofoil efficiencies (C_L/C_D) for twisted NACA0009 and NACA0015 are 49% and 46%, respectively, while for half-twisted NACA0009 and NACA0015, the efficiencies are 57% and

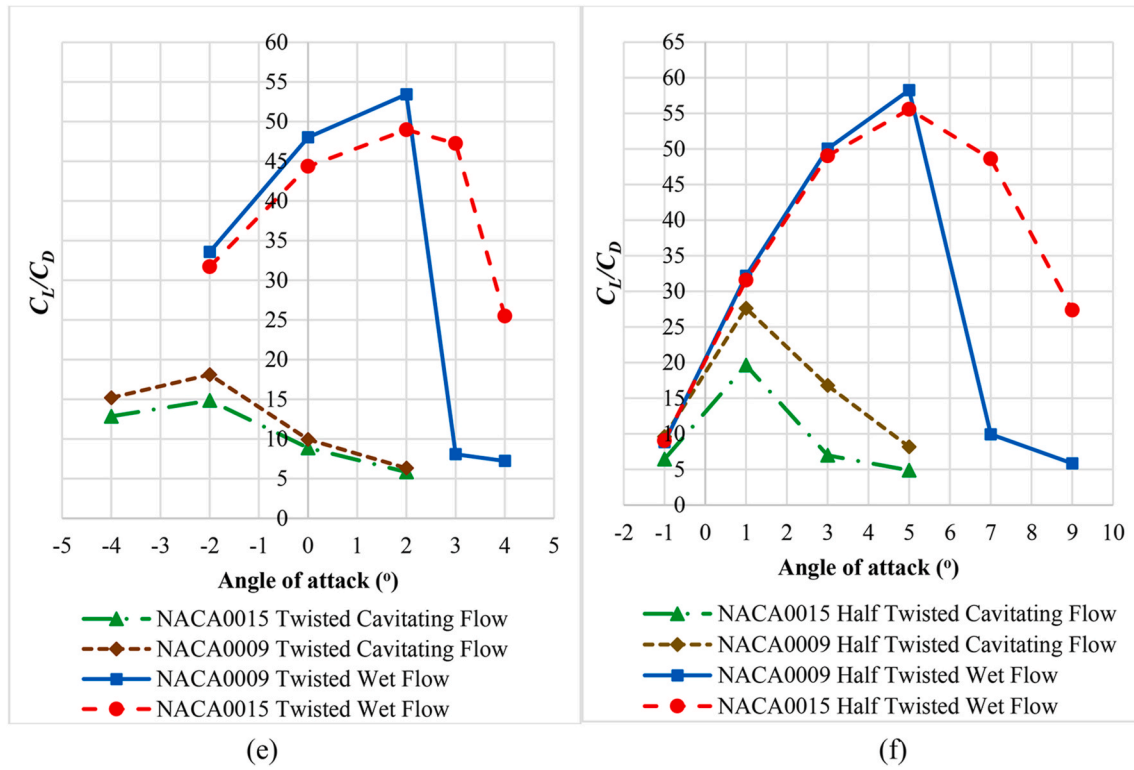
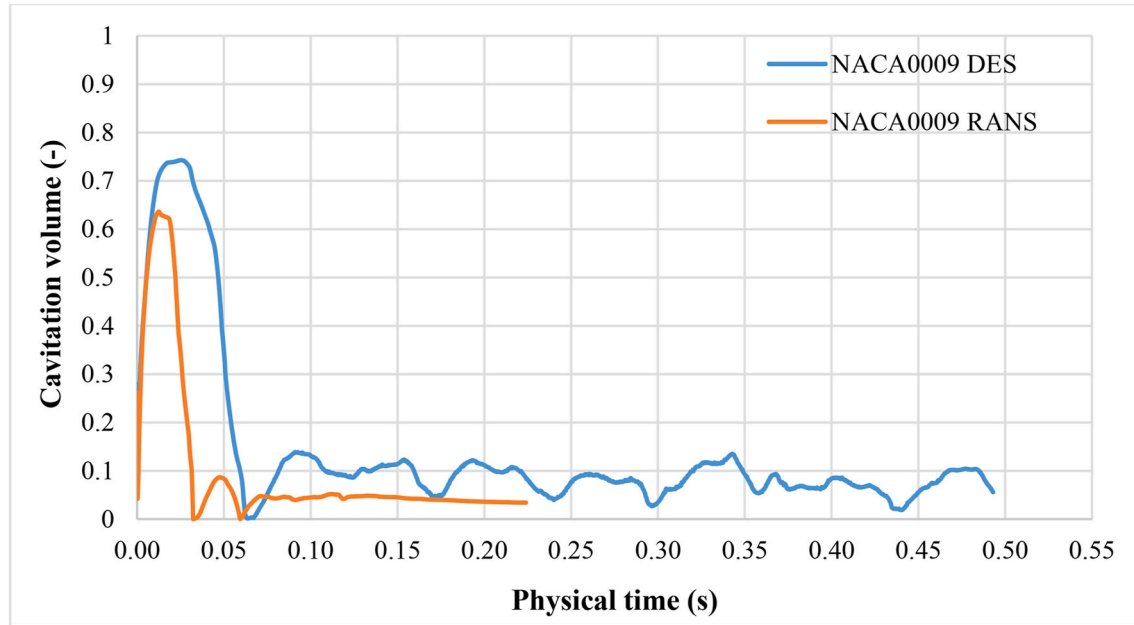


Fig. 9. (continued).

Fig. 10. Cavitation development by time for NACA0009 Twisted at $AoA = -2^\circ$ (RANS and DES).

53%, respectively. Accordingly, the highest efficiency is obtained from half-twisted NACA0009 in the non-cavitating condition. Similarly, the efficiencies of twisted NACA0009 and NACA0015 hydrofoils in the cavitating condition are 10% and 7%, whereas the efficiencies of half-twisted NACA0009 and NACA0015 hydrofoils are 12% and 5%, respectively. As a result of the analyses, it can be concluded that the efficiency increases with the decrease in twisting and section thickness, and avoidance of cavitation.

4.3. Unsteady cavitation analyses by RANS and DES methods for delft hydrofoil (NACA0009 twisted $AoA = -2^\circ$)

In this section, cavitating flow analyses of the twisted hydrofoil with NACA0009 section at -2° , with a flow velocity of $U = 6.75$ m/s and $\sigma = 1.07$, were conducted using RANS and DES methods. Since the validation section provides comparative data for lift and drag coefficients against experimental data and literature values, the variations in cavitation volume, as well as changes in lift and drag coefficients over time,

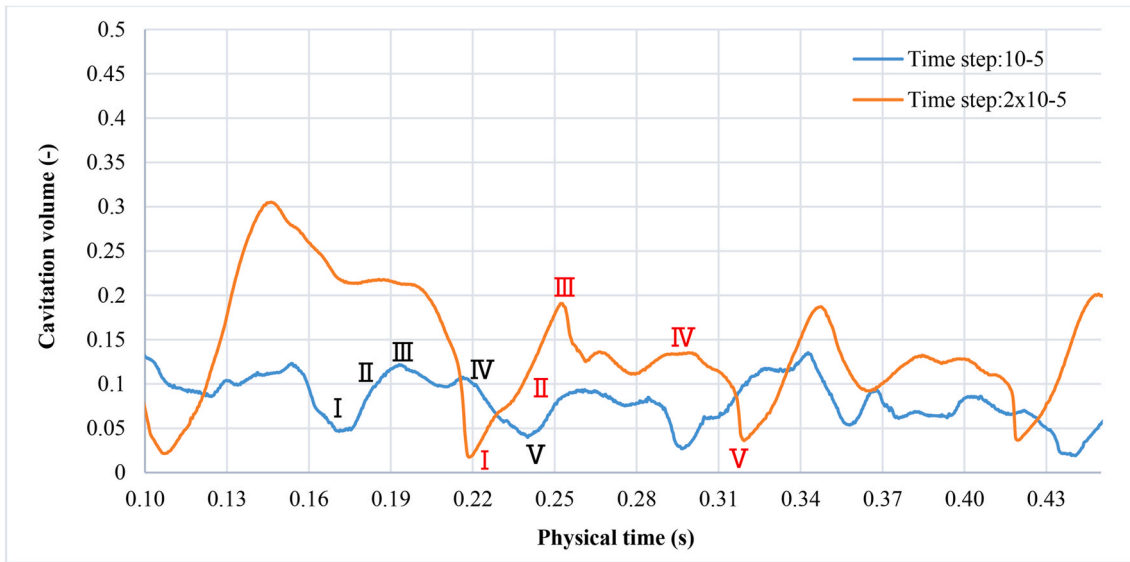


Fig. 11. Cavitation volume curves for time step $\Delta t = 1 \times 10^{-5}$ s and $\Delta t = 2 \times 10^{-5}$ s.

are presented here. Additionally, DES analyses were performed with a time step of $\Delta t = 2 \times 10^{-5}$ s, in addition to $\Delta t = 10^{-5}$ s, as detailed in subsequent subsections, to investigate the compatibility of grid density and time step in capturing cavitation volume formation within each cavitation cycle and predicting forces.

The cavitation volume shown in Figs. 10, 11, 15, 18 and 22 is a dimensionless parameter representing the ratio of vapour volume to the total volume within the domain surrounding the hydrofoil. It refers to the ratio of vapour (bubble) volume to the total volume (water + vapour).

Fig. 10, illustrates the ratio of cavitation volume to total volume over time, as calculated using DES and RANS methods with a time step of $\Delta t = 10^{-5}$ s.

While the curve derived from RANS shows a nearly constant trend after the convergence of the solution, indicating a steady state, the curve from DES exhibits distinct cavitation cycles and periods, with noticeable peaks and troughs repeating approximately. The average cavitation volume in the DES solution is slightly higher compared to RANS. Each cycle represents the development and reappearance of cavitation around the hydrofoil.

In Figs. 11–13, the temporal progression of the cavitation volumes, lift coefficient, and drag coefficient on the hydrofoil obtained from the

DES method are presented for various time intervals.

In Fig. 11, points I, II, III, IV, and V are taken as 5 physical times within one cavitation cycle, considering the time step values. Point I represents the starting point of the cavitation cycle, denoted as $T = 0$, while point V represents the completion of the cavitation cycle, denoted as $T = T$ time. Points II, III, and IV respectively represent $T/4$, $T/2$, and $3T/4$ times within the cycle.

In analyses conducted using the small-time step ($\Delta t = 1 \times 10^{-5}$ s), the cavitation cycle extends from point I at physical time $t = 0.176$ s to point V at physical time $t = 0.24$ s, resulting in a duration of approximately 0.064 s for one cavitation cycle. This cycle remains approximately consistent throughout the analysis. In analyses conducted using the higher time step ($\Delta t = 2 \times 10^{-5}$ s), the cavitation cycle extends from point I at physical time $t = 0.22$ s to point V at physical time $t = 0.32$ s, resulting in a duration of approximately 0.1 s for one cavitation cycle. Similar to the smaller time step analysis, this duration repeats itself throughout the analysis. Thus, it is possible to say that as the time step value increases, the duration of the cavitation cycle also increases.

As observed in Fig. 11, an increase in the time-step leads to an increase in both cavitation volume and period. This could be attributed to the inability to accurately capture the time-dependent cavitation formation at higher time steps.

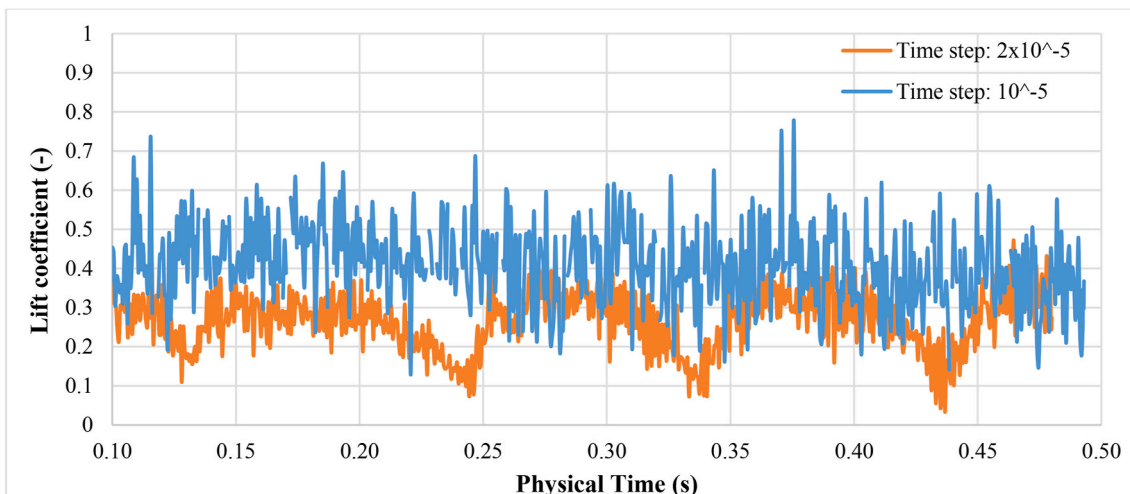


Fig. 12. Lift force coefficient curves for the NACA0009 twisted hydrofoil with $AoA = -2^\circ$, with two different time step values, $\Delta t = 10^{-5}$ s and $\Delta t = 2 \times 10^{-5}$ s.

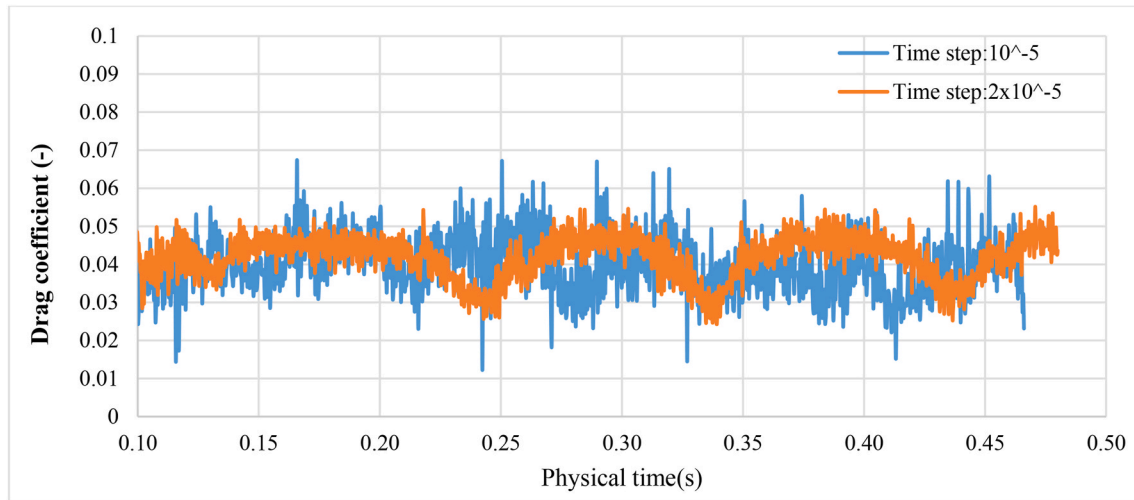


Fig. 13. Drag force coefficient curves for the NACA0009 twisted hydrofoil with $AoA = -2^\circ$, with two different time step values, $\Delta t = 10^{-5}$ s and $\Delta t = 2 \times 10^{-5}$ s.

Twisted NACA0009 $AoA = -2^\circ$	I $T=0$	II $T/4$	III $T/2$	IV $3T/4$	V T
Experiment (Foeth,2008a)					
Present Study DES Time Step: 1×10^{-5}					
Present Study DES Time Step: 2×10^{-5}					

Fig. 14. Cavitation development by time on the NACA0009 Twisted hydrofoil.

The validation section demonstrated the accuracy of the solution for a time step of $\Delta t = 10^{-5}$ s. However, the characteristics of the cycles during the cavitation period appear to be approximately similar. Furthermore, it can be observed from the lift and drag force coefficient graphs in Fig. 12 and in Fig. 13 that the periods of cavitation formation are similar to those depicted in the aforementioned figures. However, while the average drag force values at different time interval values are approximately equal ($C_D = 0.04$), it is observed that the average lift force value decreases slightly with an increase in the time step ($C_L = 0.443$ for $\Delta t = 10^{-5}$ s and $C_L = 0.28$ for $\Delta t = 2 \times 10^{-5}$ s).

In general conclusion of Figs. 12 and 13, as the time step increases, the precision of the analysis decreases, making the cavitation cycle more pronounced in the results. The same analyses were also conducted for a

time step of $\Delta t = 5 \times 10^{-5}$ s and it was observed that, the cavitation cycle became even more pronounced. However, due to significant discrepancies between the experimental and numerical results, results with a time step of $\Delta t = 5 \times 10^{-5}$ s have not been presented in this study.

In Fig. 14, images of cavitation volume within a cavitation cycle obtained from the DES method are presented comparatively with experimental images for different time steps. As can be observed from the figure, the images obtained from one cycle (at physical time $t = 0.176$ s– 0.24 s) for a time step of $\Delta t = 10^{-5}$ s are consistent with the experimental results, whereas the cavitation formations obtained from one cycle (at physical time $t = 0.22$ s– 0.32 s) for $\Delta t = 2 \times 10^{-5}$ s, exhibit slightly higher cavitation development compared to the experimental data.

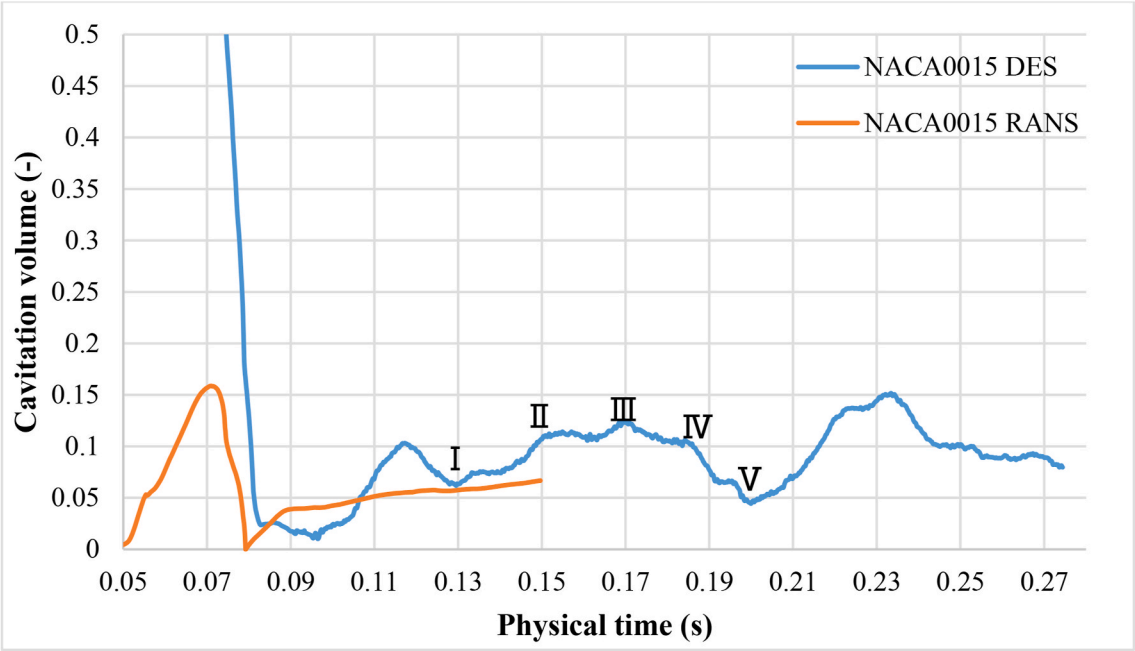


Fig. 15. Cavitation development by time for NACA0015 Twisted hydrofoil at AoA = -2° (RANS and DES).

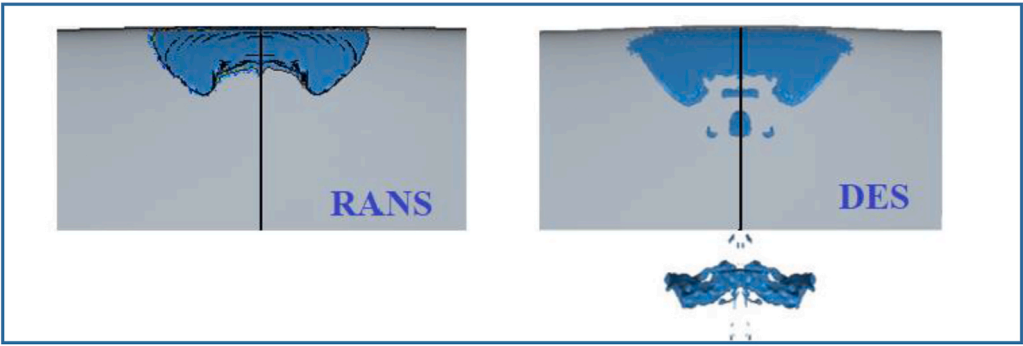


Fig. 16. Cavitation development on the NACA0015 twisted hydrofoil by RANS and DES.

DES Time Step: 10^{-5}	I T=0	II T/4	III T/2	IV 3T/4	V T
Twisted NACA0015 AoA= -2°					
Twisted NACA0009 AoA= -2°					

Fig. 17. Cavitation volume scenes over the NACA0015 and the NACA0009 twisted hydrofoils along a cycle.

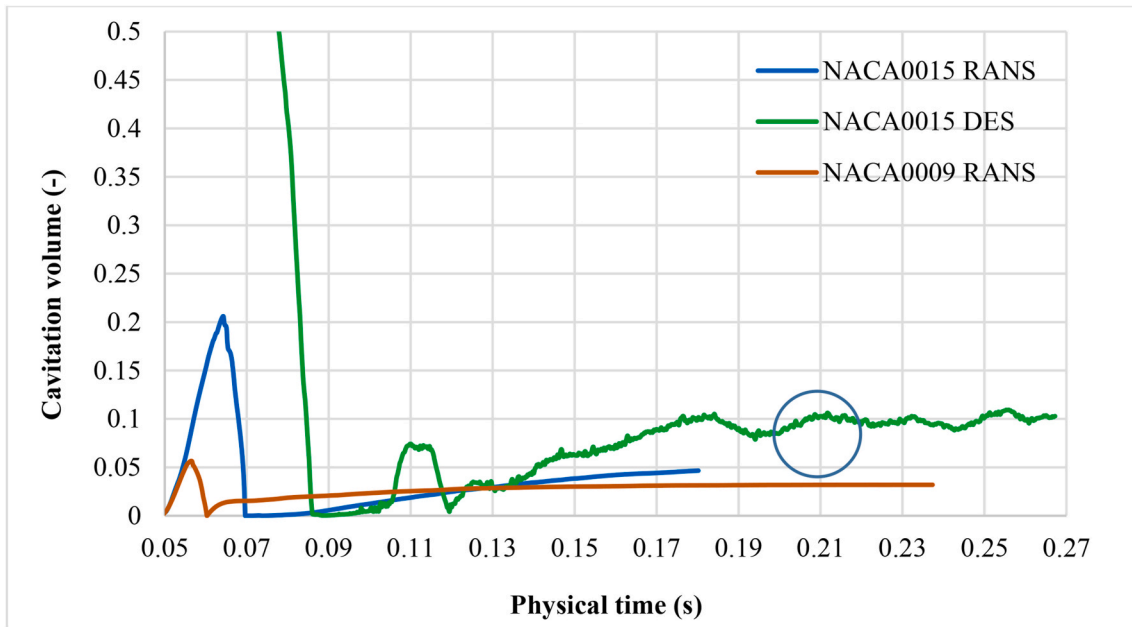


Fig. 18. Cavitation volume curves of the half-twisted NACA0009 and NACA0015, at $AoA = +1^\circ$.

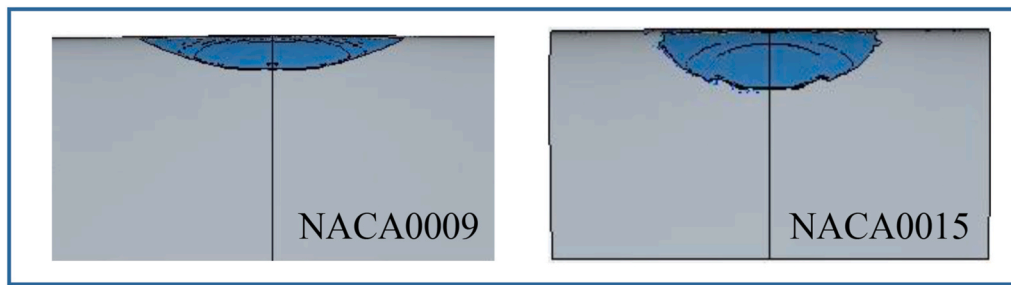


Fig. 19. Cavitation volume images over the half-twisted NACA0009 and NACA0015, at $AoA = +1^\circ$ (RANS).

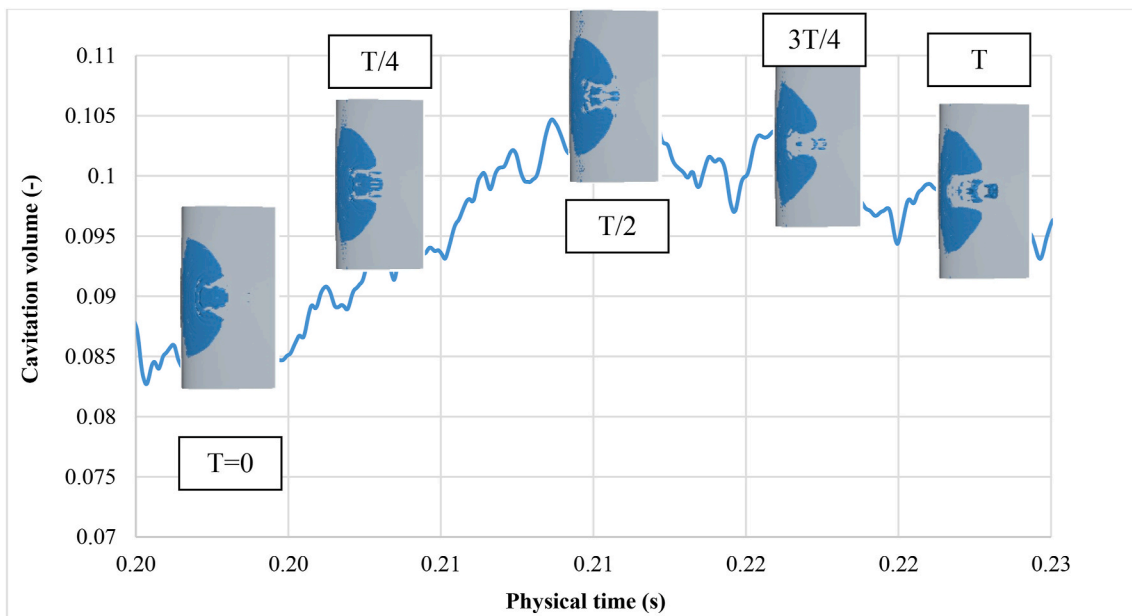


Fig. 20. Cavitation volume scenes over the NACA0015 half-twisted hydrofoil along a cycle.

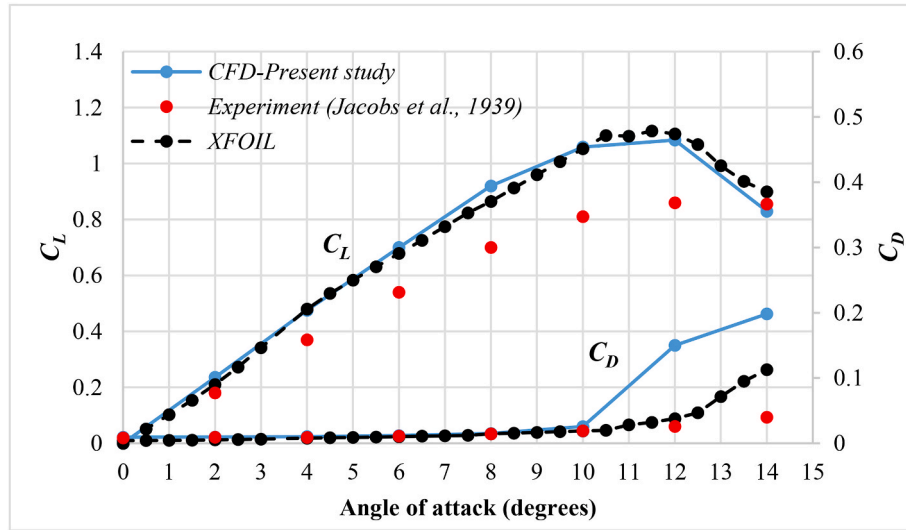


Fig. 21. Lift and drag coefficients of no twist NACA0009 depending on AoA.

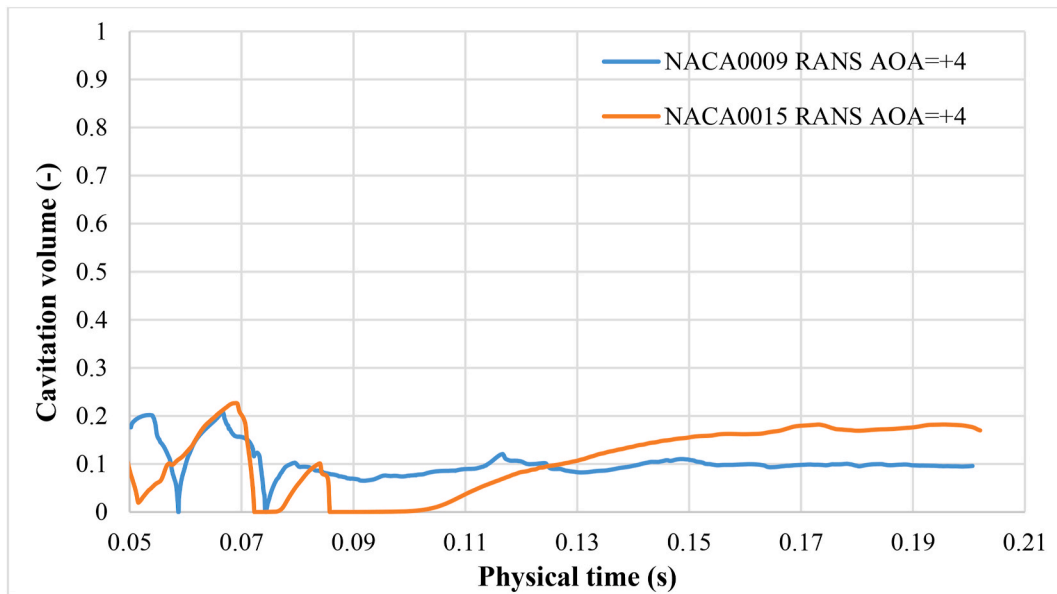


Fig. 22. Cavitation volume curves of the no-twist NACA0009 and NACA0015 at AoA = +4°.

4.4. Unsteady cavitation analyses by RANS and DES methods for delft hydrofoil (NACA0015 twisted for AoA = -2°)

In this section, the cavitating and non cavitating flow analyses of the twisted hydrofoil with NACA0015 section were performed by RANS and DES methods for time step $\Delta t = 10^{-5}$ s under the same conditions as the NACA0009 Twisted in the previous section 4.2 (AOA = -2°, $\sigma = 1.07$, inlet velocity $U = 6.75$ m/s).

Fig. 15 shows the time dependent variations of the cavitation volume around the twisted hydrofoil obtained by DES and RANS for time step $\Delta t = 10^{-5}$ s. In the curve obtained from DES, the cavitation cycle and period can be clearly seen from the approximately repeated bumps and troughs. The average cavitation volume is slightly higher in the DES solution than RANS, as it is similar to that in NACA0009 Twisted. Fig. 15 shows the cavitation formation image obtained from both methods. Although the cavitation images are quite similar for both methods, the RANS method does not sufficiently capture the cloud cavitation behind the sheet cavitation.

Similar with the NACA0009 results in Fig. 10, NACA0015 analysis

using the RANS method shows that the cavitation volume does not change after a certain analysis time, while the DES method gives a clear periodicity of vapour volume change over the hydrofoil that is observed due to the nature of cavitation. The cavitation cycle (at physical time $t = 0.128$ s– 0.198 s) for NACA0015 is found approximately 0.07s similar with the NACA0009.

Cavitation development on the NACA0015 twisted hydrofoil by RANS and DES methods are presented in Fig. 16.

In the DES method, flow characteristics and consequently cavitation development are obtained with greater precision. The lift and drag force coefficients of Twisted NACA0015 Hydrofoil in the cavitating and non-cavitating flow conditions are given as comparison those of Twisted NACA0009 in Table 4. In the analysis conducted using the RANS method, cavitation formation has been observed only near the leading edge of the hydrofoil. However, in the DES method, greater amount of cavitation formation has been observed at the leading edge of the blade, besides the cloud cavitation formation occurs at the trailing edge.

The values of lift and drag force of twisted NACA0015 and twisted NACA0009 hydrofoils are approximately the same in the non-cavitating

Table 4

Lift and drag coefficients of Twisted NACA0015 and NACA0009 hydrofoils at -2° AoA.

		Twisted NACA0015		Twisted NACA0009	
		C_L	C_D	C_L	C_D
Wetted Flow	RANS	0.438	0.0138	0.436	0.0130
Cavitation	RANS	0.308	0.0195	0.428	0.0236
	DES	0.334	0.0372	0.443	0.0401

case. However, in the case of cavitating analysis, the lift coefficients obtained from both RANS and DES are about 25% less for the thicker section twisted NACA0015. In Fig. 17, the cavitation volume images obtained from the DES method for twisted NACA0015 and twisted NACA0009 sections within one cavitation cycle are given comparatively. Fig. 17 shows that the sheet cavitation on the surface of the twisted NACA0015 is less than that of the twisted NACA0009, but from Figs. 10 and 15 it can be seen that the volume of cavitation around the thick-sectioned NACA0015 hydrofoil is greater.

4.5. Unsteady cavitating flow analyses by RANS and DES methods for delft hydrofoil (half-twisted NACA0009 and NACA0015, AoA = $+1^\circ$)

In this section, non-cavitating and cavitating flow analyses of half-twisted hydrofoils with NACA0015 and NACA0009 sections were performed by using RANS method, with a time step of $\Delta t = 10^{-5}$ s. These analyses were performed under identical conditions to the twisted analyses in sections 4.3 and 4.4 (AOA = $+1^\circ$, inlet flow velocity $U = 6.75$ m/s and $\sigma = 1.07$). Additionally, for NACA0015 half-twisted hydrofoils, flow simulations were carried out using the DES method under the same conditions.

The time dependent cavitation volume around NACA0009 half-twisted hydrofoil using the RANS method and NACA0015 half-twisted hydrofoil using DES and RANS methods are presented in Fig. 18. The average cavitation volume obtained from the RANS method is slightly higher for the NACA0015 hydrofoil. Similarly, the average cavitation volume for the NACA0015 twisted hydrofoil is slightly higher in the DES solution compared to RANS, resembling that of the NACA0015. Fig. 19 illustrates the cavitation formation on the surfaces of hydrofoils obtained from RANS. It can be observed that the cavitation formations are approximately similar (also seen in Fig. 18.)

The lift and drag force coefficients of half-twisted hydrofoils in the cavitating and non-cavitating flow conditions from DES and RANS methods are given in Table 5.

The lift and drag force coefficient values for half-twisted NACA0015 and half-twisted NACA0009 hydrofoils are observed to be approximately equal in the non-cavitating flow scenario. Similarly, the force coefficients obtained from RANS in the analysis of cavitating flow show similar values. However, the lift coefficients of the half-twisted NACA0015 hydrofoil obtained from DES exhibit slight differences compared to the RANS results.

Fig. 20 presents images of cavitation volume within a cavitation cycle (seen with circled region in Fig. 17) obtained from the DES method for a half-twisted NACA0015 hydrofoil section. As depicted in the figure, cavitation formation within a cavitation cycle remains relatively constant for the half-twisted hydrofoil at AoA = $+1^\circ$. However, for twisted

Table 5

Lift and drag coefficients of the half-twisted NACA0009 and NACA0015, at AoA = $+1^\circ$.

		NACA0015		NACA0009	
		C_L	C_D	C_L	C_D
Wetted Flow	RANS	0.346	0.0109	0.339	0.0105
Cavitation	RANS	0.345	0.0124	0.304	0.0111
	DES	0.226	0.0136	–	–

hydrofoils at a -2° angle of attack, the development and recurrence of cavitation are more clearly observed. This observation is further corroborated by the time-dependent cavitation volume variation graphs on the hydrofoils (Figs. 10, 15 and 20).

4.6. Unsteady cavitation analyses by RANS methods for delft hydrofoil (no-twist NACA0009 and NACA0015, AoA = $+4^\circ$)

In this section, non-cavitating and cavitating flow analyses of no-twist 3D NACA0009 and NACA0015 hydrofoils were conducted under the same conditions as the previously presented twisted and half-twisted hydrofoil analyses (AoA = $+4^\circ$, inlet flow velocity $U = 6.75$ m/s and $\sigma = 1.07$) using RANS methods with a time step of $\Delta t = 10^{-5}$ s.

Fig. 21 shows the comparison of lift and drag coefficients for no twist 3D NACA0009 hydrofoil at non-cavitating flow conditions depending on angle of attack; obtained from RANS method in this study, XFOIL (for Re = 1 million and Ncrit = 9) and experimental results for Re = 1.3×10^6 (Jacobs and Sherman, 1939).

Non-cavitating flow analyses for the NACA0009 hydrofoil (AR = 2) were performed under the same circumstances as for the calculation of the Delft Twisted 11 Hydrofoil in the Sub-section of 3.3 Validation of the non-cavitating flow simulations (Case 1), for angles of attack 0° – 14° , inlet flow velocity $U = 6.75$ m/s, Re = 1.14×10^6 .

XFOIL is a potential based linear-vorticity panel method with Karman-Tsien compressibility correction (Drela, 1989), which was validated with the experimental data (Cruz et al., 2022). This method is widely used in the literature for comparison of numerical results of hydrofoils.

Fig. 21 illustrates a strong agreement between CFD and XFOIL lift coefficient results up to approximately 6° angle of attack. Higher degrees of angle of attack, while not identical, the results continue to show reasonable similarity. Experimental data deviate slightly from both CFD and XFOIL predictions between 4° and 12° , consistently producing lower lift coefficients. The experimental data used in this study originated from experiments conducted in the 1930s (Jacobs and Sherman, 1939). The precision of drag and lift measurements were likely limited due to the experimental tools and techniques available at the time. For this reason, when the experimental values are analyzed, the difference especially in the lift coefficient values is an expected result. Nevertheless, all three methods predict stall to occur between 11° and 12° . Drag coefficient curves also align well for angles of attack up to around 10° .

The temporal variations of cavitation volumes on the hydrofoil surfaces are depicted in Fig. 22, while the cavitation formation images are provided in Fig. 23. From Fig. 23 (a) and 23 (b), the contour images of volume fraction of the no twist NACA0009 and NACA0015 are presented at the mid-section of the hydrofoils. As evident from Fig. 23 (a) and 23 (b), the shape of cavitation onset on both hydrofoils appears as a straight strip near the leading edge of section. From the cavitation volume curves in Fig. 22 and the cavitation formation images in Fig. 23, it can be seen that the average cavitation area and volume around the surface of the no-twist NACA0015 hydrofoil are slightly larger. The growth of cavitation volume with increasing section thickness has been similarly observed for twisted and half-twisted hydrofoils in previous analyses in this study. It is generally observed that as the section thickness of a cavitating hydrofoil increases, the cavitation volume and cavitation lengths generally tend to decrease (Kinnas, 1991; Chau et al., 2004). However, some experimental studies have shown that cavitation formation can vary differently depending on the section shape, thickness, angle of attack and cavitation number, and the tendency of cavitation formation increases as the thickness ratio increases (Fujii et al., 2007; Sarraf et al., 2007).

As can be seen from Figs. 22 and 23, when the angle of attack is $+4^\circ$, more cavitation occurs around the NACA0015 hydrofoil, which has a thicker cross section. Although this is an unexpected result, as mentioned above, geometrical or physical conditions can vary differently cavitation formation.

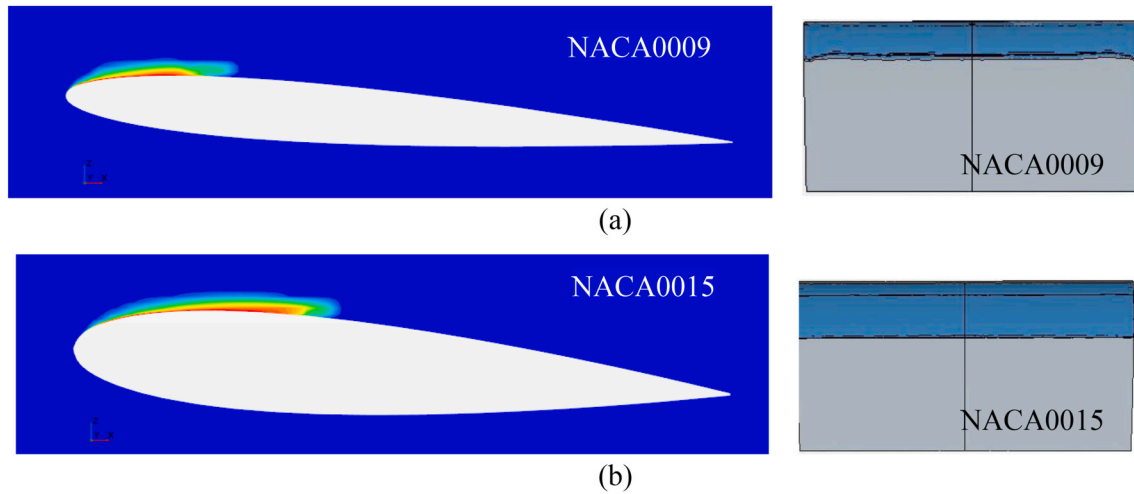


Fig. 23. 2-D volume fraction contours and 3-D cavitation development images of (a) NACA0009 hydrofoil and (b) NACA0015 hydrofoil. (no-twist NACA0009 and NACA0015, at $\text{AoA} = +4^\circ$ (RANS)).

Table 6 provides the lift and drag force coefficients of the no-twist hydrofoils under cavitating and non-cavitating flow conditions obtained from RANS methods. From the table, it is observed that the increase in hydrofoil section thickness has little influence on the lift force coefficient in the non-cavitating condition; however, it leads to a slight decrease in the cavitating condition. For the NACA0009 section hydrofoil, cavitation formation causes a minimal increase in the lift force coefficient, whereas for the other section, it results in a slight decrease.

4.7. Discussion on the numerical solution approaches for the cavitating flow analysis of hydrofoils

In this study, the cavitating flow dynamics around NACA0015 and NACA0009 hydrofoils, including twisted and half-twisted configurations, were investigated using RANS and DES methods. Significant differences between the outcomes obtained from the two methodologies were observed. With the RANS method, lift and drag coefficients were obtained in close agreement with literature values. However, the cavitation formation on the hydrofoil surface was found to be less pronounced compared to experimental images. This discrepancy suggests that RANS may not fully capture the extent of cavitation formation observed in experiments.

In contrast, DES simulations achieved results that closely matched experimental observations, particularly in terms of cavitation patterns and dynamics. Furthermore, while RANS simulations showed relatively stable cavitation formations over time, DES analyses revealed variations in cavitation dynamics, consistent with temporal changes observed in real experiments. In other words, in DES analyses cavitation formation was obtained as a periodic phenomenon along a cycle, just as it occurs in real cavitating flows.

It is also important to acknowledge the limitations of both DES and RANS methods. RANS, while efficient and capable of predicting overall hydrodynamic performance, may struggle to accurately capture transient and unsteady phenomena such as cavitation dynamics. However, while the DES method is more effective in resolving transient turbulent structures and capturing cavitation dynamics, it is computationally

more expensive, especially for high Reynolds number flows and complex geometries. In both RANS and DES methods, mesh density and time step are critical parameters. However, in DES methodology, a denser mesh is required compared to RANS, and especially the mesh around the boundary layer must be of sufficient quality.

In conclusion, while the RANS method accurately predicts lift and drag coefficients, it may not provide the same level of detail in capturing cavitation dynamics as DES. On the other hand, DES offers a more detailed representation of cavitation formation and dynamics with more total grid elements and longer analysis time.

5. Conclusions

The study comprehensively investigated the performance characteristics and time-dependent cavitation formation of twisted hydrofoils in cavitating flow conditions by RANS and DES methods, as well as their performance in non-cavitating conditions by RANS. An effort has been made to understand the impact of hydrofoil twisting (twisted, half-twisted, and no twist), angle of attack, and section thickness on hydrofoil performance and cavitation formation by varying numerical solution parameters such as the total number of grid elements in the computational volume, time step, and discretization schemes. Some of the results obtained from the analysis can be listed as follows:

- Both DES and RANS methods offer valuable insights into cavitation flow dynamics, but it is essential to recognize their respective limitations. By acknowledging these limitations and discussing their implications, researchers can better interpret simulation results and guide future developments in computational fluid dynamics methodologies.
- The steady RANS solution gives sufficient accuracy in the prediction of twisted hydrofoil performance in non-cavitating flow, while the unsteady RANS method can give sufficient results at time step value of $\Delta t = 10^{-5}$ s, in the prediction of twisted hydrofoil performance in cavitating flow. However, the RANS method cannot fully capture cavitation formation, especially cloud cavitation, and the cavitation period cannot be obtained accurately with RANS.
- In the case of cavitating flow analysis for the same geometry and under the same conditions, the cavitation volume is always larger in DES analysis than in RANS, as expected.
- In order to accurately capture the cavitation cycle and the stages of cavitation formation within a cycle around the hydrofoil, the unsteady DES method should be used at time step value of $\Delta t = 10^{-5}$ s. In RANS solutions, the number of total grids about 4.5 million and

Table 6

Lift and drag coefficients of the no-twist hydrofoils at $\text{AoA} = +4^\circ$.

	NACA0009		NACA0015	
	C_L	C_D	C_L	C_D
Wetted Flow	0.495	0.0102	0.496	0.0148
Cavitation	0.510	0.0220	0.480	0.0228

the first order discretization scheme is sufficient, while the DES method requires about 12 million grid elements and the second order discretization schemes, for the investigated conditions.

- As the angle of attack and twistiness of hydrofoil increase, it becomes difficult to capture time-dependent cavitation characteristics. So, this challenge can be overcome by using many more grid elements and much smaller time steps.
- For the twisted hydrofoil NACA0015, the stages of cavitation development within a period are obvious, whereas in the half-twisted one the image of cavitation is almost constant depending on time within a cycle.
- In the case without cavitation, the variation in the twistiness and section thickness of the hydrofoil has almost no effect on the lift, especially at small angles of attack conditions.
- More reduction in lift for all hydrofoils with NACA0015 section is seen than those of NACA0009, and increasing the twistiness of hydrofoil causes more notable effect.
- In cavitation analysis, increasing the time step in DES method makes the cavitation development cycle more visible, especially in the lift and drag force graphs, but it is insufficient to accurately estimate the lift force coefficient and the cavitation images at stages within a period. The cavitation period also increases approximately in proportion to the time step.

As a future study the impact of twisting geometry under different cavitation conditions for various hydrofoil sections, with the aim of examining the erosion prediction within these variations is planned to be investigated.

CRedit authorship contribution statement

Sinem Öksüz: Writing – review & editing, Writing – original draft, Visualization, Validation, Software, Methodology, Investigation, Formal analysis, Data curation. **Onur Usta:** Writing – review & editing, Validation, Supervision, Software, Methodology. **Fahri Celik:** Writing – review & editing, Validation, Supervision, Methodology, Investigation.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgements

The authors are grateful to Batuhan AKTAS, who is the lecturer in the NAOME Department at the University of Strathclyde, for his assistance and providing workstation permissions during the studies within Erasmus + programme. The authors would like to thank to University of Strathclyde Naval Architecture and Marine Engineering Department for provision of the ARCHIE-WeSt high performance computing facilities.

This research has been financially supported by Yildiz Technical University Research Grants (Grant FBA-2024-6032).

References

- Arabnejad, M.H., Amini, A., Farhat, M., Bensow, R.E., 2020. Hydrodynamic mechanisms of aggressive collapse events in leading edge cavitation. *J. Hydrodyn.* 32 (1), 6–19. <https://doi.org/10.1007/s42241-020-0002-8/METRICS>.
- Chau, S.W., Hsu, K.L., Kouh, J.S., Chen, Y.J., 2004. Investigation of cavitation inception characteristics of hydrofoil sections via a viscous approach. *J. Mar. Sci. Technol.* 8, 147–158. <https://doi.org/10.1007/s00773-003-0165-3>.
- Chen, Y., Li, J., Gong, Z., Chen, X., Lu, C., 2019. Large eddy simulation and investigation on the laminar-turbulent transition and turbulence-cavitation interaction in the cavitating flow around hydrofoil. *Int. J. Multiphas. Flow* 112, 300–322. <https://doi.org/10.1016/j.ijmultiphaseflow.2018.10.012>.
- Cheng, H., Long, X., Ji, B., Peng, X., Farhat, M., 2021. A new Euler-Lagrangian cavitation model for tip-vortex cavitation with the effect of non-condensable gas. *Int. J. Multiphas. Flow* 134. <https://doi.org/10.1016/j.ijmultiphaseflow.2020.103441>.
- Cruz, A., Rodriguez, A., Kotteda, V.M.K., Paez, B., Villanueva, D., Chijioke, C., Terrazas, J., Kumar, V., 2022. Aerodynamic Performance of Symmetric and Cambered Airfoils at Low Reynolds Numbers. 1. American Society of Mechanical Engineers, Fluids Engineering Division (Publication) FEDSM. <https://doi.org/10.1115/FEDSM2022-86960>.
- Dongli, M., Yanping, Z., Yuhang, Q., Guanxiong, L., 2015. Effects of relative thickness on aerodynamic characteristics of airfoil at a low Reynolds number. *Chin. J. Aeronaut.* 28 (4), 1003–1015. <https://doi.org/10.1016/J.CJA.2015.05.012>.
- Drela, M., 1989. XFoil: an analysis and design system for low Reynolds number airfoils. Low Reynolds number aerodynamics. In: Mueller, t.j. (Ed.), *Proc. Conf., Notre Dame, U.S.A., June 5-7, 1989, 1989*. Springer-Verlag, Berlin, Germany, pp. 1–12. https://doi.org/10.1007/978-3-642-84010-4_1 (Lecture notes in, 54).
- Ferziger, J.H., Peric, M., Street, R., 2020. Computational Methods for Fluid Dynamics. Springer International Publishing. <https://doi.org/10.1007/978-3-319-99693-6>.
- Foeth, E.J., 2008a. The Structure of Three Dimensional Sheet Cavitation. Delft University of Technology, Department of Marine Technology (PhD Thesis).
- Foeth, E.J., 2008b. Pressure and Lift Measurements on Twist-11 Hydrofoil. Delft University of Technology (Report).
- Fujii, A., Kawakami, D.T., Tsujimoto, Y., Arndt, R.E.A., 2007. Effect of hydrofoil shapes on partial and transitional cavity oscillations. *J. Fluid Eng.* 129 (6), 669–673. <https://doi.org/10.1115/1.2734183>.
- Ghasemnezhad, M., Roohi, E., 2024. Large eddy simulation of cavitating flow around a pitching hydrofoil. *Ocean. Eng.* 292, 116547. <https://doi.org/10.1016/J.OCEANENG.2023.116547>.
- Gritskevich, M.S., Garbaruk, A.v., Schütze, J., Menter, F.R., 2012. Development of DDES and IDDES formulations for the k- ω shear stress transport model. *Flow, Turbul. Combust.* 88 (3), 431–449. <https://doi.org/10.1007/S10494-011-9378-4/METRICS>.
- He, P., Li, Z., Liu, Q., Zhang, X., He, W., 2023. Prediction of cavitation dynamics and cavitation erosion around a three-dimensional twisted hydrofoil with an LES method. *Int. J. Nav. Archit. Ocean Eng.* 15, 100536. <https://doi.org/10.1016/J.IJNAOE.2023.100536>.
- Hoekstra, M., Terwisga, T.V., Foeth, E.J., 2011. Smp'11 Workshop Cavitation on Foil, - Case 1: Delft Foil. Second International Symposium on Marine Propulsors smp'11, Hamburg, Germany. June 2011. <http://maritimetechnology.tudelft.nl/SHS/>.
- Jacobs, E.N., Sherman, A., 1939. Airfoil section characteristics as affected by variations of the Reynolds number. [https://doi.org/10.1016/S0016-0032\(37\)90818-4](https://doi.org/10.1016/S0016-0032(37)90818-4).
- Ji, B., Luo, X., Arndt, R.E.A., Wu, Y., 2014. Numerical simulation of three dimensional cavitation shedding dynamics with special emphasis on cavitation-vortex interaction. *Ocean. Eng.* 87 (November 2017), 64–77. <https://doi.org/10.1016/j.oceaneng.2014.05.005>.
- Ji, B., Luo, X.W., Arndt, R.E.A., Peng, X., Wu, Y., 2015. Large Eddy Simulation and theoretical investigations of the transient cavitating vortical flow structure around a NACA66 hydrofoil. *Int. J. Multiphas. Flow* 68, 121–134. <https://doi.org/10.1016/J.IJMULTIPHASEFLOW.2014.10.008>.
- Kim, J.H., Choi, J.E., Choi, B.J., Chung, S.H., 2014. Twisted rudder for reducing fuel-oil consumption. *Int. J. Nav. Archit. Ocean Eng.* 6 (3), 715–722. <https://doi.org/10.2478/IJNAOE-2013-0207>.
- Kinnas, S.A., 1991. Leading-edge corrections to the linear theory of partially cavitating hydrofoils. *J. Ship Res.* 35, 15–27. <https://doi.org/10.5957/jsr.1991.35.1.15>.
- Koop, A.H., 2008. Numerical Simulation of Unsteady Three-Dimensional Sheet Cavitation. University of Twente. <https://doi.org/10.3990/1.9789036527019>. PhD Thesis.
- Köksal, Ç.S., Usta, O., Aktas, B., Atlar, M., Korkut, E., 2021. Numerical prediction of cavitation erosion to investigate the effect of wake on marine propellers. *Ocean. Eng.* 239, 109820. <https://doi.org/10.1016/J.OCEANENG.2021.109820>.
- Kumar, P., Singh, S., Reddy, S.R., 2022. Assessment of unsteady cavitating flow around modified NACA4412 hydrofoil. *Ocean. Eng.* 266, 113039. <https://doi.org/10.1016/J.OCEANENG.2022.113039>.
- Li, D.-Q., Grekula, M., Lindell, P., 2009. A modified SST k- ω Turbulence model to predict the steady and unsteady sheet cavitation on 2D and 3D hydrofoils. Proceedings of the 7th International Symposium on Cavitation CAV2009. <https://repository.tudelft.nl/islandora/object/uuid%3A6478c3de-4960-420d-848f-16805d4dd25c>.
- Liu, T., Kuykendoll, K., Rhew, R., Jones, S., 2006. Avian wing geometry and kinematics. *AIAA J.* 44 (5), 954–963. <https://doi.org/10.2514/1.16224>.
- Liu, H.L., Wang, J., Wang, Y., Zhang, H., Huang, H., 2014. Influence of the empirical coefficients of cavitation model on predicting cavitating flow in the centrifugal pump. *Int. J. Nav. Archit. Ocean Eng.* 6 (1), 119–131. <https://doi.org/10.2478/IJNAOE-2013-0167>.
- Liu, D.C., Hong, F.W., 2010. The numerical analysis of unsteady performance of cavitating flow on 3D hydrofoils and comparison with experiments. *Chinese Journal of Hydrodynamics* 25 (6), 721–726.
- Liu, H., Guo, Q., Shi, L., Tang, F., Dai, L., Shen, J., Liu, J., 2023. Lift-drag characteristics of S-shaped hydrofoil under different cloud cavitation conditions. *Ocean. Eng.* 278, 114374. <https://doi.org/10.1016/J.OCEANENG.2023.114374>.
- Li, Z.R., 2012. Assessment of Cavitation Erosion with a Multiphase Reynolds-Averaged Navier-Stokes Method. PhD. Thesis. TU Delft, Delft. ISBN 9789065623003. <https://repository.tudelft.nl/islandora/object/uuid%3Aa23364b6-de5f-481a-8a33-25a9aede01c5>.
- Long, X., Cheng, H., Ji, B., Arndt, R.E.A., Peng, X., 2018. Large eddy simulation and Euler-Lagrangian coupling investigation of the transient cavitating turbulent flow around a twisted hydrofoil. *Int. J. Multiphas. Flow* 100, 41–56. <https://doi.org/10.1016/j.ijmultiphaseflow.2017.12.002>.
- Miki, T., Fujiyama, K., Tsuda, S.-I., Irie, T., 2021. Cavitation analysis of a Delft twisted hydrofoil using multi-process cavitation model. *J. Phys. Conf.* 1909 (1). <https://doi.org/10.1088/1742-6596/1909/1/012014>.

- Melissaris, T., Bulten, N., van Terwisga, T.J.C., 2019. On the applicability of cavitation erosion risk models with a URANS solver. *Journal of Fluids Engineering, Transactions of the ASME* 141 (10). <https://doi.org/10.1115/1.4043169>.
- Movahedian, A., Pasandidehfar, M., Roohi, E., 2019. LES investigation of sheet-cloud cavitation around a 3-D twisted wing with a NACA 16012 hydrofoil. *Ocean. Eng.* 192 (March) <https://doi.org/10.1016/j.oceaneng.2019.106547>.
- Oksuz, S., Celik, F., Bayraktar, S., 2023a. Three-dimensional computational analysis of flow over twisted hydrofoils. *Ocean. Eng.* 267, 113304 <https://doi.org/10.1016/J.OCEANENG.2022.113304>.
- Oksuz, S., Usta, O., Celik, F., 2023b. Numerical investigation of the twisted profile effect on hydrofoils in cavitating flow conditions. In: *Proceedings of 4th International Naval Architecture and Maritime Symposium*, Ytu, Istanbul, pp. 11–13, 2023.
- Paik, K.J., Park, H.G., Seo, J., 2013. RANS simulation of cavitation and hull pressure fluctuation for marine propeller operating behind-hull condition. *Int. J. Nav. Archit. Ocean Eng.* 5 (4), 502–512. <https://doi.org/10.2478/IJNAOE-2013-0149>.
- Pant, C.S., Frankel, S.H., 2021. Interaction between surface blowing and re-entrant jet in active control of hydrofoil cavitation. *Ocean. Eng.* 242, 110087 <https://doi.org/10.1016/j.oceaneng.2021.110087>.
- Pendar, M.R., Alavi, A., Roohi, E., 2023. Identification of Frequency Modes and Spectral Content for Noise Suppression: Cavitation Flow over 3-D Hydrofoil with Sinusoidal Leading Edge, 34. <https://doi.org/10.1142/S0129183123500742>, 6.
- Sarraf, C., Djeridi, H., Billard, J.-Y., 2007. Thickness and Cavitation Effects on Vibrations of Hydrofoils at Large Angle of Attack, 10. https://doi.org/10.1007/978-1-4020-8630-4_20i.
- Seeni, A., 2022. A theory on understanding aerodynamic phenomena of airfoils and the significance of airfoil's thickness on lift and drag. *INCAS Bulletin* 14 (3).
- Shi, Y., Luo, K., Chen, X., Li, D., Jia, L., 2021. A new cavitation model considering inter-bubble action. *Int. J. Nav. Archit. Ocean Eng.* 13, 566–574. <https://doi.org/10.1016/J.IJNAOE.2021.05.005>.
- Shur, M.L., Spalart, P.R., Strelets, M.K., Travin, A.K., 2008. A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities. *Int. J. Heat Fluid Flow* 29 (6), 1638–1649. <https://doi.org/10.1016/J.JHEATFLUIDFLOW.2008.07.001>.
- Sogukpinar, H., 2017. Numerical simulation of 4-digit inclined NACA 00xx airfoils to find optimum angle of attack for airplane wing. *Uludag University Journal of The Faculty of Engineering* 22 (Issue 1). <https://doi.org/10.17482/uumfd.309470>.
- Soto, C., Bhattacharya, S., 2023. The effect of dynamic twisting on the flow field and the unsteady forces of a heaving flat plate. *Bioinspiration Biomimetics* 18 (2), 026010. <https://doi.org/10.1088/1748-3190/ACB7BA>.
- Star CCM, 2022. User Guide. Siemens.
- Usta, O., 2018. Modelling of Cavitation and Cavitation Erosion on Marine Propellers (Gemi Pervanelerinde Kavitasyon Ve Kavitasyon Erozyonu Modellemesi, in Turkish). Istanbul Technical University. *Ph.D. Thesis*.
- Usta, O., Korkut, E., 2018. A study for cavitating flow analysis using DES model. *Ocean. Eng.* 160, 397–411. <https://doi.org/10.1016/J.OCEANENG.2018.04.064>.
- Usta, O., Korkut, E., 2019. Prediction of cavitation development and cavitation erosion on hydrofoils and propellers by Detached Eddy Simulation. *Ocean. Eng.* 191, 106512 <https://doi.org/10.1016/j.oceaneng.2018.04.064>.
- Vaz, G., Lloyd, T., Gnanasundaram, A., 2017. Improved modelling of sheet cavitation dynamics on Delft Twist11 hydrofoil. In: *MARINE VII: Proceedings of the VII International Conference on Computational Methods in Marine Engineering*. CIMNE, pp. 143–156.
- Wang, Z., Li, L., Li, X., Yang, C., Zhu, Z., 2023a. Cavitation evolution around a twist hydrofoil by large eddy simulation (LES) with mesh adaption. *J. Ocean Univ. China* 22 (3), 627–636. <https://doi.org/10.1007/S11802-023-5208-2/METRICS>.
- Wang, Z., Cheng, H., Ji, B., Peng, X., 2023b. Numerical investigation of inner structure and its formation mechanism of cloud cavitating flow. *Int. J. Multiphas. Flow* 165, 104484. <https://doi.org/10.1016/J.IJMULTIPHASEFLOW.2023.104484>.
- Whitworth, S., 2011. Cavitation prediction of flow over the Delft twist 11 foil. In: *Second International Symposium on Marine Propulsors*, pp. 15–17.
- Wilcox, D.C., 2006. *Turbulence Modelling for CFD*, third ed. DCW Industries, L a Canada CA <http://www.dcwindustries.com>.
- Wu, X.C., Wang, Y.W., Huang, C.G., 2016. Effect of mesh resolution on large eddy simulation of cloud cavitating flow around a three dimensional twisted hydrofoil. *Eur. J. Mech. B Fluid* 55, 229–240. <https://doi.org/10.1016/j.euromechflu.2015.09.011>.
- Xie, N., Tang, Y., Meng, L., Wei, Y., 2023. High-fidelity numerical simulation of unsteady cavitating flow around a hydrofoil. *J. Hydrodyn.* 35 (1), 1–16. <https://doi.org/10.1007/S42241-023-0014-2/METRICS>.
- Xu, R.E., Wu, Z., 2023. Numerical simulation of flow over airfoil and its optimization. *J. Phys. Conf.* 2441 (1) <https://doi.org/10.1088/1742-6596/2441/1/012004>.
- Yin, T., Pavesi, G., Pei, J., Yuan, S., 2021. Numerical investigation of unsteady cavitation around a twisted hydrofoil. *Int. J. Multiphas. Flow* 135. <https://doi.org/10.1016/j.ijmultiphaseflow.2020.103506>.
- Yin, R., Liu, H., He, Z.-Y., 2022. Numerical simulation of hydrofoil cavitation. *J. Phys. Conf.* 2206 (1) <https://doi.org/10.1088/1742-6596/2206/1/012002>.
- Zhang, D., Chen, J., Shi, L., Zhang, G., Shi, W., van Esch, B.P.M., 2018. Numerical analysis of the unsteady cavitation shedding flow around twisted hydrofoil based on hybrid filter model. *Therm. Sci.* 22 (4), 1629–1636. <https://doi.org/10.2298/TSCI1804629Z>.
- Zhang, G., Wang, Z., Wu, C., Li, H., Sun, T., 2023. Numerical investigation of multistage cavity shedding around a cavitating hydrofoil based on different turbulence models. *Ocean. Eng.* 284, 115248 <https://doi.org/10.1016/J.OCEANENG.2023.115248>.