Proceedings of the 8th World Congress on Momentum, Heat and Mass Transfer (MHMT'23)
Lisbon, Portugal – March 26 – 28, 2023
Paper No. ICMFHT 122
DOI: 10.11159/icmfht23.122

Simulating Cloud Cavitation Using Detached Eddy Simulation and Other Hybrid Turbulence Models

Dhruv Apte[1], Mingming Ge[1,2], Olivier Coutier-Delgosha[1,3]
[1]Kevin T. Crofton Department of Aerospace and Ocean Engineering, Virginia Tech,
Blacksburg, VA-24060, USA
dhruvga@vt.edu; mmge@vt.edu
[2]National Energy Technology Laboratory, Morgantown, WV 26507, USA
[3]Univ. Lille, CNRS, ONERA, Arts et Metiers ParisTech, Centrale Lille, FRE 2017 -
LMFL -Laboratoire de Mecanique des fluides de Lille - Kampe de Feriet, F-59000 Lille, France
ocoutier@vt.edu

Abstract **-** Cavitation is defined as the formation of bubbles resulting from a sharp drop in vapor pressure at a near constant liquid temperature. Both numerical and experimental methods are needed to study this phenomenon in detail. On the numerical side, the cavitation-turbulence coupling in the process makes cavitating flows difficult to model. One particular approach, using a homogeneous Transport-Equation Model (TEM) coupled with a turbulence model has been utilized in this study where the influence of a hybrid RANS-LES turbulence model on the turbulence properties and its ability to behave as an LES model properly is investigated. The Detached Eddy Simulation (DES), Delayed DES (DDES) and Improved DDES (IDDES) are coupled with the Merkle cavitation model to simulate cloud cavitating flow inside a venturi using interPhaseChangeFoam, a multiphase flow solver in OpenFOAM. Comparisons are made for void fraction and turbulence properties on the global and local scale by comparing them with high-fidelity data obtained from X-ray Particle Image Velocimetry (PIV) experiments.

Keywords**:** Cavitation, OpenFOAM, turbulence modelling, hybrid RANS-LES models

1. Introduction

A complex phenomenon characterized by the formation of vapour bubbles when the liquid pressure drops the vapour pressure at a near-constant liquid temperature, cavitation has received great attention due to its strong background in engineering applications like propellers, pumps and turbine blades. When these bubbles are transported to the high-pressure region, the bubbles collapse causing high frequency pressure oscillations and shocks inducing stress, vibrations, erosion damage and load asymmetry that have a detrimental effect on the efficiency of the above-mentioned engineering applications. Previously, various cavitation studies have been carried out on various geometries experimentally and using numerical simulations.

Modelling cavitating flow involves using efficient techniques to model the cavitation-turbulence coupling. Cavitation can either be modelled as a sharp interface that separates the liquid and vapour or using averaged sets of fluid dynamics equations with the number of equations ranging from seven to only three. A notable mention is the Transport Equation Model (TEM) which assumes a homogeneous mixture of fluid and vapour connected by the void transport equation. The transport equation utilizes separate terms for condensation and evaporation terms which could be treated separately for analysis. Various TEM models have been formulated to model cavitating flows[1, 2, 3, 4]. From the perspective of the turbulence modelling approach, some studies use Reynolds-Averaged Navier-Stokes (RANS) models where the Navier-Stokes equations are averaged and all the scales are modelled while some studies use Large Eddy Simulations (LES) that resolves turbulent scales upto a filter length and models the smaller scales. But LES is computationally costly and thus most studies use RANS models due to its low computational costs, good stability and acceptable accuracy. RANS models have been unable to reproduce the periodic vapor shedding of cavitating flows and over-estimate the turbulent eddy viscosity [5]. When compared to experimental data on a local scale, they show large discrepancies on comparing the turbulence properties like Reynolds shear stress and turbulent kinetic energy. [6]. Another class of turbulence models, termed as hybrid RANS-LES models have been recently used to model cavitating flows. This class of model uses a LES model away from the wall and a

RANS model near the wall, thus modelling more accurate flow dynamics while also scaling down the grid size and subsequently, the computational costs. Several studies have been conducted to compare the performance of hybrid RANS-LES models with RANS models and experimental data for simulating cavitation. Ponkratov & Caldas [7] used a Detached Eddy simulation (DES) along with erosion functions to predict cavitation erosion near the leading edge of a rudder caused by a ship propeller. Although the model could predict cavitation on a global scale, it under-predicted the size of cavity and the area of the collapsing region although it could predict the erosion damage accurately. Sedlar et al. [8] compared cavitating flow over a hydrofoil using experiments, the k-ω SST-SAS (Scale-Adaptive Simulations), DES and LES models. While LES indeed could predict more vortical structures than the other two models, both models were able to predict the decrease of cavity length to zero and seemed to describe the cavity break-up better than the SAS model. They also stated that both LES and DES model prediction could be influenced by the computational grids. Additionally, the sudden switch from LES to RANS as we move towards the wall in DES results in a flow-induced separation. Therefore, Spalart et al. [9] introduced Delayed DES (DDES) model. Long et al. [10] used the DDES model to predict cavitating flow for a hydrofoil and a marine propeller behind the hull, followed by a verification and validation study. They concluded that the DDES model predicts the transient flow accurately in general but is not able to predict the local flow structures and the large-scale vortices. It also confirmed that the model is sensitive to grids as the error difference between experimental and numerical studies grows smaller and more structures are captured. However, these above-mentioned studies have not compared the turbulence properties like the Reynolds wall shear and the turbulent kinetic energy. Comparing these properties, especially on the local scale would give further insights if these models are able to predict the turbulence coupled with cavitation. The goal of this work is to assess the ability of DES and other related models like the DDES and the IDDES models to predict cavitation by comparing with experimental data on both global and local scale.

The rest of the paper is as follows: In Section 2, we discuss the numerical setup. Section 3 presents the results along with a written analysis and concludes the paper.

2. Numerical Setup

The computational geometry is a converging-diverging nozzle (a *venturi*) from the experiments of Ge et al. [11]. It has a 18 degree convergent and 8 degree divergent angle. The sharp angle sat the throat enable the flow to be turbulent. The height of the venturi is 21 mm but it reduces to 11 mm at the throat. The inlet velocity is set to 8.38 m/s while the pressure is adjusted to meet the mean cavity length as seen in the experiments. Probes have been placed at 1.5, 3, 5, 10, 15 and 20 mm from the throat for local scale analysis.

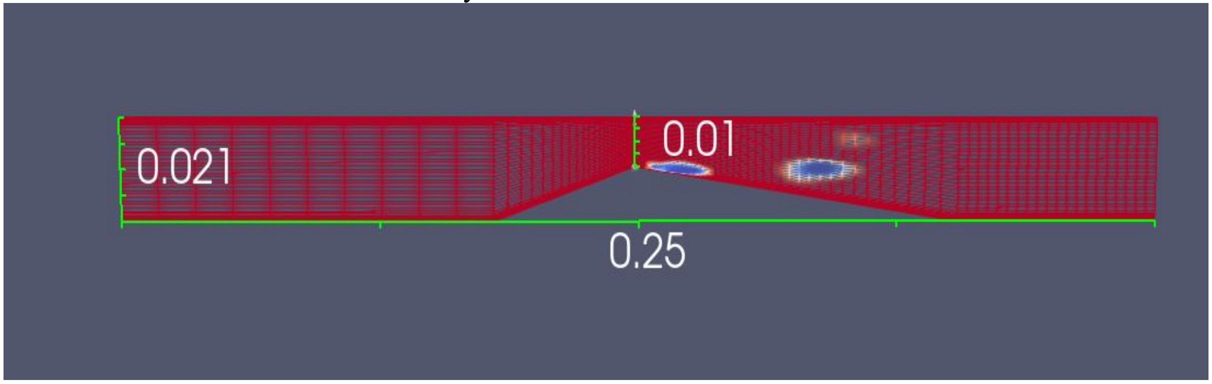


Figure 1: The venturi geometry used in the study

The Merkle cavitation model [1] from the Transport-Equation Model family is selected as the cavitation model. It has separate terms for both vapor formation (evaporation) and vapor destruction (condensation) with some empirical constants. We first define the continuity and momentum equations:

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m u_j)}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial(\rho_m u_i)}{\partial t}+\frac{\partial(\rho_m u_i u_j)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[(\mu_m+\mu_t)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)\right] \tag{2}$$

$$\frac{\partial \rho_l \alpha_l}{\partial t}+\frac{\partial(\rho_l \alpha_l u_j)}{\partial x_j}=\dot{m}^{+}+\dot{m}^{-} \tag{3}$$

Where ρ_m, μ_m are the mixture phase density and viscosity respectively while u and p being the mixture velocity and mixture pressure respectively; $\rho_l, \rho_v, \mu_l, \mu_v$ being the liquid density, vapour density, liquid dynamic viscosity and vapor dynamic viscosity respectively. μ_t represents the turbulent eddy viscosity while α_l and α_v are the liquid and vapor void fraction respectively. The subscripts *(i,j,k)* represent the Cartesian coordinate system

The terms $\dot{m}^{+}$ and $\dot{m}^{-}$ represent the vapor formation and destruction terms respectively. They are given by:

$$\dot{m}^{+}=\frac{C_{prod}\max(p-p_{sat},0)(1-\gamma)}{0.5U_\infty^2 t_\infty} \tag{4}$$

$$\dot{m}^{-}=\frac{C_{dest}\min(p-p_{sat},0)\gamma\rho_l}{0.5\,U_\infty^2 t_\infty \rho_v} \tag{5}$$

Where γ is the liquid volume fraction, p and p_{sat} are the pressure and saturation pressure respectively while t_∞ and U_∞ are the free stream time scale and free stream velocity respectively. The empirical constants C_{des} and C_{prod} are set as 1e-3 and 80 respectively.

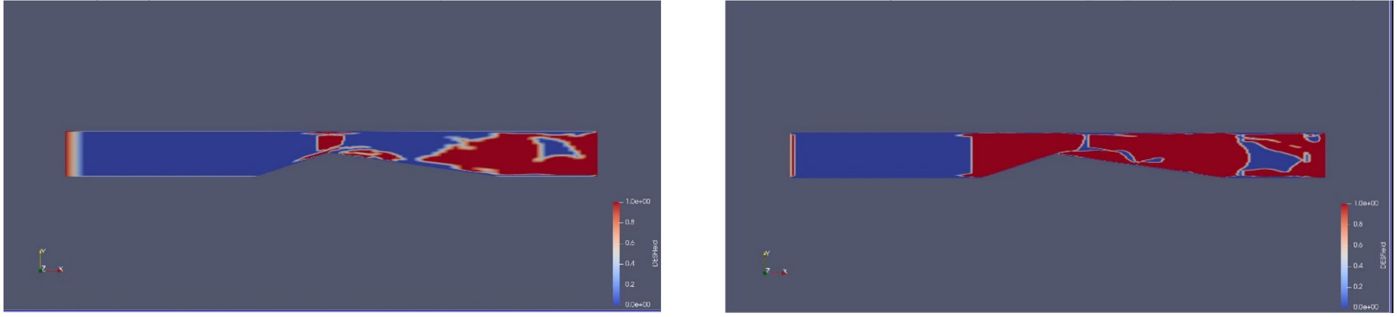
The geometry is meshed in OpenFOAM [12], an open-sourced platform composed of a multitude of solvers. The solver used in this study is an isothermal, unsteady, multiphase solver titled *interPhaseChangeFoam.* To understand the dependence of grid refinement on the DES model family, three meshes have been designed. The meshes have been designed to ensure more flow region, especially the cavity region is modelled by LES rather than RANS (see Table 1). Initially, a fully turbulent, non cavitating flow regime is run for 0.03s followed by a sinusoidal ramp for another 0.03s. During this ramp, there is no cavitation at the start while at the end of the ramp, there is full cavitation. Following this, a fully cavitating regime is launched for 1s. The focus of our results is on the final 1s of fully cavitating flow. The timestep used throughout this study is 1e-5.

Table 1- Grids used in the study

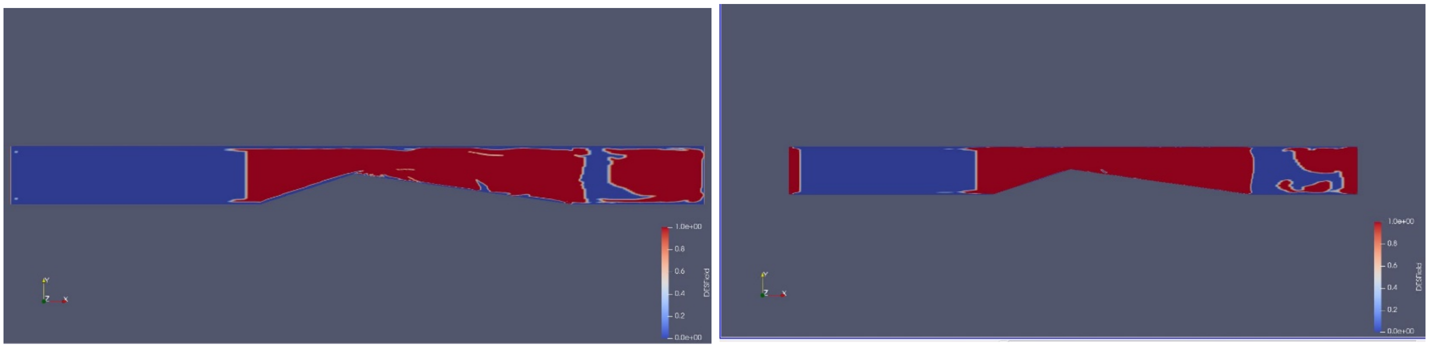
Grid Number	Grid Size
1	35,500
2	51,600
3	84,000

3. Results and discussion

Before going into the local scale analysis, it is crucial to know if the LES region is modelled properly over the cavity region.



(a) (b)



(c) (d)

Figure 2 LES modelling zones regions in four different meshes- a) 5,000 cells b) Grid 1 c) Grid 2 d) Grid 3. The red zone indicates the zone is modelled by LES while the blue region indicates RANS models the zone.

Figure 2 shows the LES modelled regions along with a previous study with the same geometry and model but on a coarser mesh [6]. Initially, with the 5,000 cells only a small part of the throat is modelled and most of the LES modelling happens downstream, ahead of the cavity formation and destruction region. With the rise in grid size, it can be observed more regions near the throat are modelled including the primary detachment zone especially from Grid 1 to Grid 2.

In the earlier study, it has been shown the unsteady vapor shedding cycles have been simulated in coarser meshes but there on the local scale, there exists a significant gap between numerical simulations and experiments. With these models having LES regions, numerical simulations can yield both modelled stress components and fluctuating stress components.

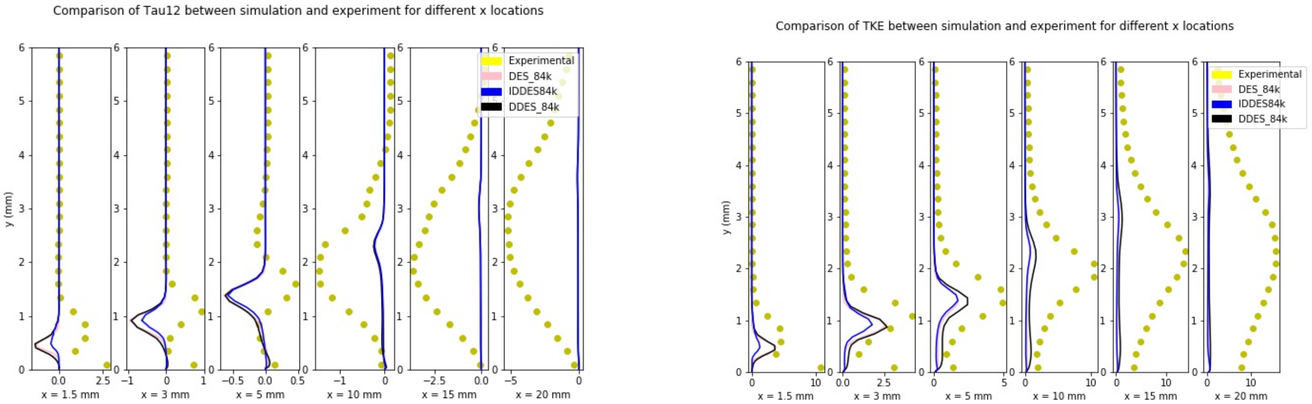


Figure 3- Modelled Reynolds wall shear stress and TKE for the three models used in this study on Grid 3

Figure 3 shows the modelled turbulence properties on a local scale for the three models on the finest grid with the experimental data (in yellow). It can be observed that none of the models are able to reproduce the turbulence on a local scale. While they are able to predict the turbulence properties near the throat up to a certain extent, they are completely unable to predict the turbulence properties downstream. It can be argued that the resolved stress components could be compared and probably able to predict the turbulence. However, on a comparison with the resolved stress components, it was observed that the resolved components are also unable to predict the turbulence properties. It is also interesting to note that despite significant improvements from the DES model by the DDES and IDDES models in theory, all three models predict a very similar behavior on the local scale.

4. Conclusion

In this study, the ability of various models of the DES family on refined grids to predict cloud cavitation in a venturi cavity is evaluated. Conducting global and local comparisons shows that these hybrid RANS-LES models are able to improve the modelling of cavitating flows up to a certain extent only. While more regions are modelled by LES on grid refinement, the improvements in regards to the turbulence properties are not substantial and it seems further grid improvements would not ameliorate these problems. Thus, there seems to be a bottleneck with hybrid models unable to reproduce cavitation and the its turbulence-coupling on the local scale. Data-driven techniques that utilize experimental/DNS data as a training dataset to devise an algorithm seem the next promising approach in predicting unsteady cavitating flows.

References

[1] Merkle, C. L. (1998). Computational modelling of the dynamics of sheet cavitation. In *Proc. of the 3rd Int. Symp. On Cavitation, Grenoble, France, 1998*.

[2] Kunz, R. F., Boger, D. A., Stinebring, D. R., Chyczewski, T. S., Lindau, J. W., Gibeling, H. J., ... & Govindan, T. (2000). A preconditioned Navier–Stokes method for two-phase flows with application to cavitation prediction. *Computers & Fluids*, *29*(8), 849-875.

[3] Sauer, J., & Schnerr, G. H. (2000, June). Unsteady cavitating flow-a new cavitation model based on a modified front capturing method and bubble dynamics. In *Proceedings of 2000 ASME fluid engineering summer conference* (Vol. 251, pp. 1073-1079).

[4] Zwart, P. J., Gerber, A. G., & Belamri, T. (2004, May). A two-phase flow model for predicting cavitation dynamics. In *Fifth international conference on multiphase flow, Yokohama, Japan* (Vol. 152).

[5] Coutier-Delgosha, O., Fortes-Patella, R., & Reboud, J. L. (2003). Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation. *J. Fluids Eng.*, *125*(1), 38-45.

[6] Apte, D., Ge, M., & Coutier-Delgosha, O. (2021). Comparison of Reynolds shear stress methods for RANS turbulence modelling of a cloud cavitation in a venturi. In *APS Division of Fluid Dynamics Meeting Abstracts* (pp. P28-008).

[7] Ponkratov, D., & Caldas, A. (2015). Prediction of cavitation erosion by detached eddy simulation (des) and its validation against model and ship scale results. In *Proceeding of the 4th International Symposium on Marine Propulsors, Austin, TX, USA* (Vol. 31).

[8] Sedlar, M., Ji, B., Kratky, T., Rebok, T., & Huzlík, R. (2016). Numerical and experimental investigation of three-dimensional cavitating flow around the straight NACA2412 hydrofoil. *Ocean Engineering*, *123*, 357-382.

[9] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. K., & Travin, A. (2006). A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theoretical and computational fluid dynamics*, *20*(3), 181-195.

[10] Long, Y., Han, C., Long, X., Ji, B., & Huang, H. (2021). Verification and validation of Delayed Detached Eddy Simulation for cavitating turbulent flow around a hydrofoil and a marine propeller behind the hull. *Applied Mathematical Modelling*, *96*, 382-401.

[11] Ge, M., Petkovšek, M., Zhang, G., Jacobs, D., & Coutier-Delgosha, O. (2021). Cavitation dynamics and thermodynamic effects at elevated temperatures in a small Venturi channel. *International Journal of Heat and Mass Transfer*, *170*, 120970.

[12] Weller, H. G., Tabor, G., Jasak, H., & Fureby, C. (1998). A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in physics*, *12*(6), 620-631.