ANALYSIS AND REDUCTION OF CAVITATION EFFECT ON SUBMERGED HYDROFOILS

Mrs. Rohini D, Ms. Madhumitha D, Mr. Karthik K, Mr. Sathis Kumar A

Department of Aeronautical engineering & Bannari Amman Institute of Technology

Abstract: Cavitation is a significant issue for the maritime sector. It has an impact on how well ships operate. The investigation of cavitation in 2D and 3D hydrofoils is the focus of this project (NACA 0015). The volume fraction of vapour reaches a value of 1 with the aid of an appendage (artificial portion above the surface), and a density decrease happens close to the surface. The result of this was a decrease in the rupture stress, bubble collapsing rate, and cavity formation size. Nonetheless, it results in a tiny increase in drag force numbers. This technique allowed for the control of erosion rate. The need determines the hydrofoil to be used. Using computational fluid dynamics, the procedure is carried out for totally turbulent flow (CFD). The grid independent study's mesh count was chosen. The chosen hydrofoil (NACA 0015) is analyzed over using pressure-based and multiphase functions in FLUENT Version 15 at a depth of 12.5 m. The drag curves and density contours are examined for three different cavitation numbers (0.5, 0.45, and 0.4), velocities (29.13 m/s, 31 m/s, and 33 m/s), and Reynolds numbers (37.32 x 10, 39.72 x 10°, and 42.3 x 10"). The appendage findings and the values of drag forces produced from the analysis (6290.7202, 9140.3964, and 16931.562) (14887.848, 13592.145 and 16955.367). The outcome of cavitation reduction is contrasted with the prior outcome. There is a comparison of the outcomes.

1.Introduction:

A lifting surface or foil that acts in water is called a hydrofoil. These resemble airfoils used by airplanes in look and function. Hydrofoils are used by boats to travel across the water. Hydrofoils raise the boat's hull out of the water as speed increases, reducing drag and enabling higher speeds. There are two persistent issues that hydrofoil designers must deal with.

- Ventilation
- Cavitation

Ventilation happens when a portion of a hydrofoil pierces the water's surface and air is drawn down the lifting surface of the foil. The foil produces far less lift and the boat crashes down because air is much less dense than water. Any air-water interaction has the potential to be ventilated.

In a liquid, vapour cavities develop by a process called cavitation. It is a result of the cavitational liquid being subjected to cavitational forces. The creation of cavities, where the pressure is low, generally happens when a liquid is exposed to rapid pressure variations. Higher pressure causes voids to collapse, which can produce a strong shock wave. In some engineering applications, cavitation is a significant source of wear. Collapsing voids that repeatedly collapsed close to a metal surface produced cyclic stress. The metal casing surfaces become fatigued as a result. Cavitation is another name for a form of wear. Pump impellers and bends, where there is an abrupt change in fluid direction, are the most typical instances of this type of wear. The rapid collapse of cavitation generates shock waves that exceed the local speed of sound in the fluid.

Fundamentals of Cavitation:

When the local pressure is dropped to a level that is close to the liquid's saturation vapors pressure, cavitation, a dynamic phenomenon, takes place in the liquid. Cavitation can be seen visually when vaporous bubbles or cavities occur. The cavitation number, which is the primary parameter of cavitation, is determined by,

$$\boldsymbol{\sigma} = \boldsymbol{Pref} - \boldsymbol{Pvop} \,/\, (\mathbf{1}/\,\mathbf{2}\ \boldsymbol{\rho V}\ \boldsymbol{ref}\ ^{2})$$

σ - Cavitation number
P (ref) - Reference pressure
P (vap) - Vapor pressure
V (ref) - Reference velocity

The Reynolds Number is one of the most crucial concepts in fluid dynamics. The relationship between inertia force and viscous force determines the Reynolds number.

$$\mathrm{Re} = \boldsymbol{\rho V D} / \mu$$

ρ - Density if the fluid
V - Velocity
D - Characteristic diameter
µ - Fluid viscosity

This equation determines if a flow will be laminar or turbulent using the Reynolds number. If the Reynolds number is high (>2100), the flow is turbulent because inertial forces predominate over viscous forces. Low Reynolds numbers (1100) indicate that viscous forces are dominant and laminar flow is occur.

2.Cavitation Inception:

Cavitation often occurs when there is an incipient case of cavitation in a constant flow over a streamlined body. It is influenced by

$$\boldsymbol{\sigma = -Cp\ (min)}$$

Vapour Pressure:

Vapor pressure or equilibrium vapor pressure is the pressure of a vapor in thermodynamic equilibrium with its condensed phases in a closed container.

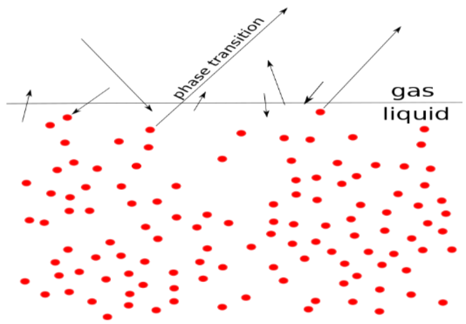


Fig.1 Vapor pressure

Boiling Point Temperature:

In a closed container, the evaporation process will continue until there are an equal number of molecules returning to the liquid as there are molecules exiting. The vapor is considered to be saturated at this stage, and the pressure of that vapor— which is often given in mmHg—is known as the saturated vapor pressure.

3. Drag:

The force that resists forward motion through the atmosphere and is perpendicular to the direction of the free-stream velocity of the flow is referred to as drag. To go ahead, force must be used to overcome drag.

4.Types Of Drag:

- Induced drag
- Parasite drag
- Form drag(Pressure drag)
- Interference drag
- Trim drag
- Skin friction drag

5. Bubble Dynamics:

Bubbles are the major issues faced in marine industries. It deals with bubble behavior such as:

- Nucleation
- Growth
- Collapse

Nucleation:

At a specific, consistent temperature, a pure liquid is said to be under tension when it is exposed to a pressure that is lower than its vapour pressure. A pure liquid may withstand extremely high negative pressures without rupturing and generating vapour holes, much like a solid, if there is no vapour present. In such condition of tension, it is possible to maintain stability.

The random thermal motion of the molecules leading to the creation of brief gaps between the molecules. Homogeneous nucleation is the name given to the method by which vapour bubbles form. At the boundary between a liquid and a solid that is in touch with the liquid, there may be additional weaknesses. We refer to this as heterogeneous nucleation.

Growth:

The Rayleigh-Plesset equation will regulate the growth and deflation of a bubble containing gas and vapour in a decreased pressure environment after it has begun. The expansion of the bubble will go rather steadily. It only continues past a certain point. When anything goes beyond that point, a bubble bursts.

Collapse:

The ultimate and most hazardous area of the bubble dynamic process is here. The bubbles clash with the surface here, eroding the surface as a result. Every bubble in this area expelled at a pressure of 2000 and a temperature of 5000 degrees Celsius. On the hydrofoil surface, it causes the cyclic strain. The structure will experience fatigue loading, which ultimately causes failure.

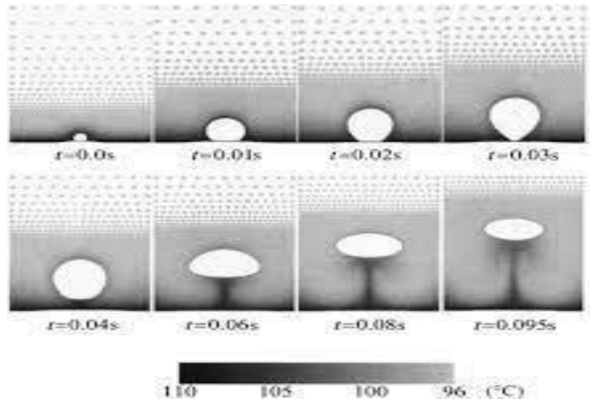


Fig.2 Bubble dynamics

6. Appendage:

It is the additional part on the surface of the hydrofoil which is used to reduce the bubble size and erosion. Its location near the growth of the cavitating bubble. It creates increasing the pressure drag and decreasing the viscous.

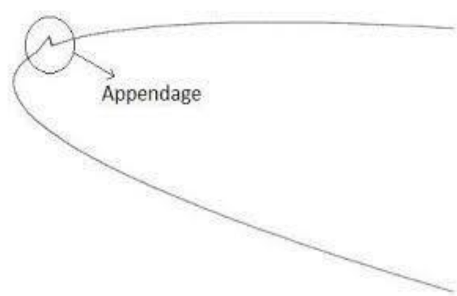


Fig .3 Schematic diagram of the appendage

7.Design Calculation and Net Generation for Hydrofoil with Ansys Fluent:

Governing Equations Of CFD:

The governing equations for a fluid are obtained by applying the fundamental rules of physics to the fluid. Along with the conservation of energy equation, the conservation of mass and momentum equations make up a set of linked, nonlinear partial differential equations. For the majority of engineering issues, these equations cannot be solved analytically. For a number of engineering issues, it is feasible to derive approximative computer-based solutions to the governing equations. The topic of computational fluid dynamics is this (CFD).

Steps to be followed in CFD:

- Surface meshing
- Volume meshing
- Solving the CFD problem
- Post processing
- Report generation

Solving the CFD Problem:

1. Reading the file. The reading the file should case file or data file or case and data file. In this we have to case and data file.
2. Scaling the grid.
3. Checking the grid.
4. Defining the modes. Model should define whether it is steady or unsteady and viscous. The model is defined here is steady and viscous.
5. Defining material.
6. Defining boundary condition
7. Controls
8. Initialize
9. Monitor
10. Iterate

Post Processing:

The element used to analyze, display, and interactively show the results is known as the post-processor. Obtaining point values and creating intricate animation sequences are both examples of post-processing. Examples of some crucial characteristics of post-processors include:

1. Geometry and control volume visualization.

2. Vector charts displaying the flow's amplitude and direction

3. Calculations that are quantitative in nature

4. Animation

5. Diagrams displaying changeable graphical plots

6. Output in hardcopy and online.

Report Generation:

All graphs, tables, figures, and comments are converted into report material automatically. The sequence of the report's components can be changed, and the figures can be bitmaps or 3D Viewer files.

8. Geometry of The Hydrofoils:

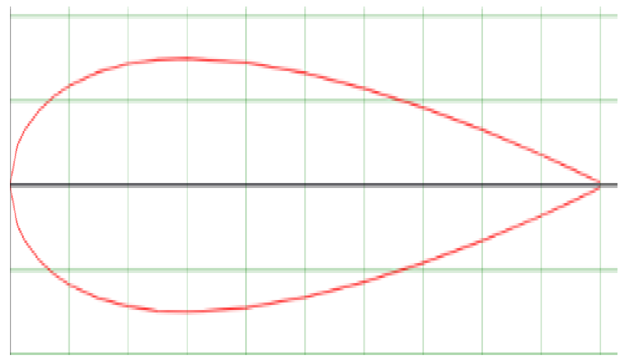


Fig.4 NACA 0015

9. Domain's Geometry:

The establishing boundary condition for the given geometry is the most crucial component of the CFD. If the boundary conditions are not properly given, the solution may be incorrect and the amount of time it takes to solve the issue may rise. Figure 1 depicts the boundary condition. The establishing boundary condition for the given geometry is the most crucial component of the CFD. If the boundary conditions are not properly given, the solution may be incorrect and the amount of time it takes to solve the issue may rise. Figure 1 depicts the boundary condition. The establishing boundary condition for the given geometry is the most crucial component of the CFD. If the boundary conditions are not properly given, the solution may be incorrect and the amount of time it takes to solve the issue may rise. Figure 1 depicts the boundary condition.

If the boundary conditions are not properly given, the solution may be incorrect and the amount of time it takes to solve the issue may rise. Figure 1 depicts the boundary condition.

10. Grid Generation for D2 Hydrofoils: without Appendage:

The FLUENT pre-processor ANSA was used to create a computational mesh. ANSA is a design programme that offers precise design and mesh. The NACA 0015 hydrofoil was the focal point of the D2 domain. The domain first experiences surface meshing, then volume meshing. A triangular mesh is available for the entire surface. Near the foil surface, fine mesh was given; elsewhere, coarse mesh was produced.

WITH APPENDAGE:

This kind of analysis follows the same steps as those used for the basic model. Appendage is the artificially sharp corner on the hydrofoil's surface that is used to reduce the size of the bubbles as a result of an abrupt decrease in velocity. The appendage model is a modified NACA 0015 model.

As a result of the symmetry of the hydrofoil we chose, the appendage is positioned on both the top and bottom sides of the hydrofoil surface.

11.Grid Generation for D3 Hydrofoils:

The FLUENT pre-processor ANSA was used to create a computational mesh. Around the NACA 015 hydrofoil, D3 domain was established. The domain first experiences a surface mesh, then a volume mesh. There is a triangular mesh available for the entire surface. Near the foil surface, fine mesh was given; elsewhere, coarse mesh was The following is a schematic drawing of a 3D hydrofoil with an 1m chord and its domain with mesh.

12. Simulation of Cavitation Using Fluent: Fluent:

For industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from blood flow to semiconductor production, and from clean room design to waste water treatment facilities, ANSYS FLUENT software has the wide physical modelling capabilities required. The software's scope has been expanded by special models that enable it to simulate in-cylinder combustion, aeroacoustics, turbo machines, and multiphase systems. This generated quick, precise CFD results. "ANSYS FLUENT 15" may be used for flow simulation.

13. Numerical Solutions:
Solver Settings:

Type	Pressure based
Velocity Formulation	Absolute
Time	Steady
2D Space	planner

Table.1 Solver options

Types:

- Absolute velocity
- Relative velocity

Pressure Based:

Every flow simulation may utilize the pressure-based approach to solve the pressure equation and conserve mass, with the exception of those that use actual gas models and non-reflecting boundary conditions. The sole option for multiphase situations is a pressure-based paradigm. So, in our research, we conduct our analysis using the pressure-based solver. "Developed primarily as a pressure-based solution for incompressible flows".

Density Based:

The initial continuity equation, together with the associated sets of equations for momentum, energy, and species transport, are all solved by the density-based technique. The following models are not compatible with this solver: the VOF model, the multiphase mixture model, the Eulerian multiphase model, the composition PDF transport model, the soot model, the Roseland radiation model, the melting/solidification model, the Shell conduction model, the floating operating pressure, the fixed variable option, the physical velocity formulation for porous media, the relative velocity formulation, and The velocity field is derived from momentum equations in both approaches. The density field is obtained using the continuity equation in the density-based approach, whereas the pressure field is derived using the equation of state.

The pressure field is extracted in the pressure-based technique, however, by solving a pressure or pressure correction equation that is created by modifying continuity and momentum equations.

14. Velocity Formulation:

- Absolute velocity
- Relative velocity

Absolute Velocity:

The application's desired absolute velocity when the flow in the dome is not revolving (Example: a fan in a large room). The absolute formulation is always employed when one of the linked solution algorithms is applied.

Regardless of whether the absolute or relative velocity is employed in the computation, we may give the velocity for the intake and outlet in either the absolute or relative frame. As our model hydrofoil is not rotating, we employ absolute velocity in our article.

RELATIVE VELOCITY:

In cases where the flow in the domain is spinning (such as flow through impeller blades), the relative velocity is most desired. Coupled solvers do not have it accessible.

15. Time:

- Steady state
- Unsteady state (Transient)

Steady State:

The steady time symbol in fluid means that the characteristics of the fluid won't vary throughout time. The outcomes won't change even if the period is extended. It will remain consistent after the findings are obtained. As unstable state requires more time and is more difficult to analyze, we utilize steady state.

Unsteady State:

It only states that the fluid's characteristics alter as time passes. The results won't be consistent; they'll alter as the time is extended.

16. 2D Space:

- 2D PLANNAR
- Axi-symmetric
- Axi-symmetric swirl

17. 2D Planar:

The full 2d geometry is used to analyze using this option. It gives the result of the d2 geometry with the absolute velocity formulation. it is not suitable of the relative velocity and rotating components. We use this option because our model is 2dhydrofoil.

18. AXIS-SYMMETRIC:

While using the original shape's half, we choose this alternative. Analyzing the substantial components is important. When one half part has been examined, the other will produce a symmetric outcome.

Axi-symmetrc Swirl:

This option is for rotating geometry, such as the impeller blades in an impeller flow. This kind is more appropriate for analysis after we apply the relative velocity concept.

19. Models:

Multiphase Model:

Any fluid flow with more than one phase or component is referred to as multiphase flow. The working medium is assumed to be a single fluid with a homogeneous mixing of phases by the multiphase mixture model FLUENT 15.0. (liquid and vapour). As a result, the mixture field's RANS equations could only be solved once.

Turbulent Modeling:

Several flow fields must be calculated using turbulent modelling. As the great majority of flows are turbulent, ANSYS FLUENT software has always placed a premium on offering cutting-edge turbulence models to properly and effectively represent the impacts of turbulence. We investigate further cavitation-related impacts on their hydrofoil using the Spalart-Almaras turbulent model. It resolves a single turbulent viscosity conservation equation (PE). It was created to be used in unstructured code in the precise aerospace sector. For wall-bounded flows, how with separation, and light recirculation, it provides the most accurate results.

Primary phase:

Seawater liquid

- Liquid density = 1025 kg/m'
- Liquid viscosity = 0.0008 kg/m-s

Secondary phase:

Seawater vapour

- Vapour density = 0.025 kg/m°
- Vapour viscosity = 0.00001 kg/m-s
- Vapour pressure = 0.04242 bar

20. Model for Cavitation:

The cavitation model in FLUENT must first be enabled. The "complete cavitation model," created by Singhal et al., serves as the foundation for this cavitation model. It explains all first-order effects (ie, phase change, bubble dynamics, turbulence fluctuations). It is able to take multi-phase flows into consideration. With or without slip velocity, the cavitation model may be used to the mixed multiphase model. It is usually better to solve cavitation, though. The issue shows that there is substantial slip between phases when the slip velocity is switched on. As gravity is irrelevant in this flow, high level turbulence prevents the formation of huge bubbles. Thus, we do not need to activate the slip velocity.

21. Results:

For a particular hydrofoil (NACA 0015), 2D and 3D analyses were performed.ANSYS Fluent version 15 was used for these studies, which were performed for three distinct cavitation values (0.5, 0.45, and 0.4). Following the investigation, the hydrofoil's surface underwent alteration. In order to modify the hydrofoil on both sides for the same cavitation numbers, we chose a symmetric hydrofoil, and the results of the analysis for this adjustment were compared to the earlier results. the analysis of the results in comparison.

22.Model Validation:

Case1: Cavitation on a Hydrofoils.

Cavitation on the surface of the 3D hydrofoil is taken into consideration as the initial test case. For analysis, the 1m symmetric 3D hydrofoil is used. When the foil is exposed to the flow of fluid, cavitation begins in the low pressure area. The computation shown below yielded the velocity values for three distinct cavitation numbers (0.5, 0.45, and 0.4). Figure illustrates the decline in density and the increasein vapour concentration. Since it is the value of mixture density, the density will notexceed the value of vapour density. In accordance with the curves representing the coefficients of lift and drag as well as the values of drag force shown in the table, the value of the volume fraction of vapour (VoF) increases with increasing velocity.

Condition: 1

For cavitation number =0.5

= (2.32-0.04242) X10^5

V =29.13m/s

Re =37.32 X10^6

Condition: 2

For cavitation number =0.4

= (2.32-0.04242) X10^5

V =33m/s

Re = 42.3 X 10^6

23. Analysis of Hydrofoil Without Appendage:

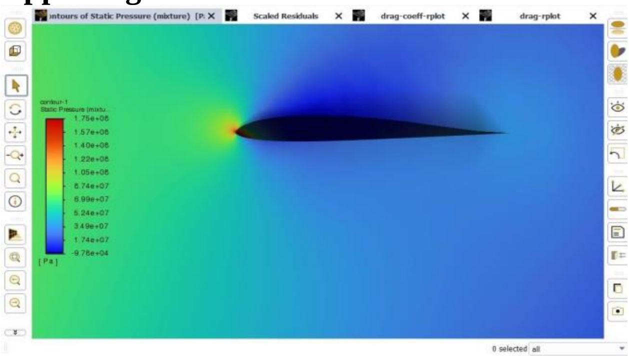


Fig.5 Pressure contour of hydrofoil without appendage

24.Velocity Counter of Traingular Appendage:

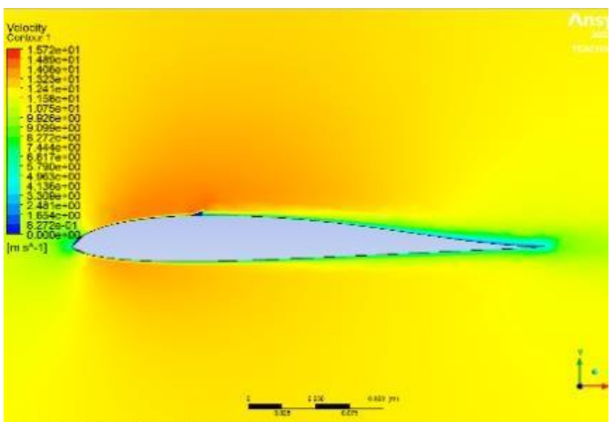


Fig.6

25.Pressure Contour of Triangular Appendage:

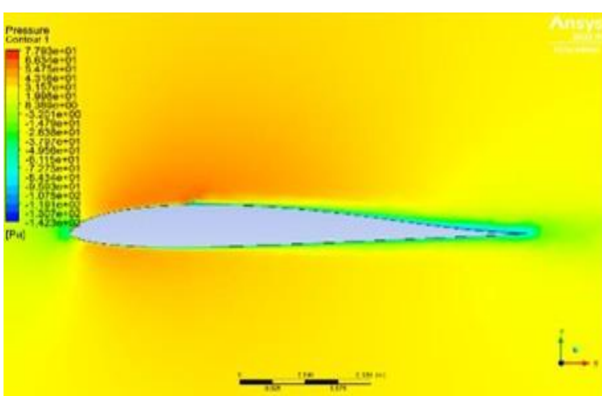
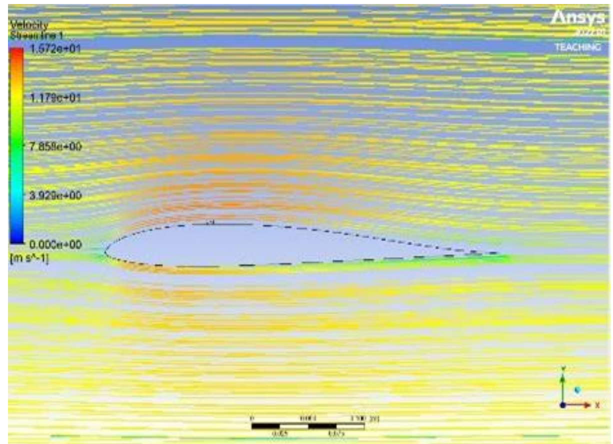
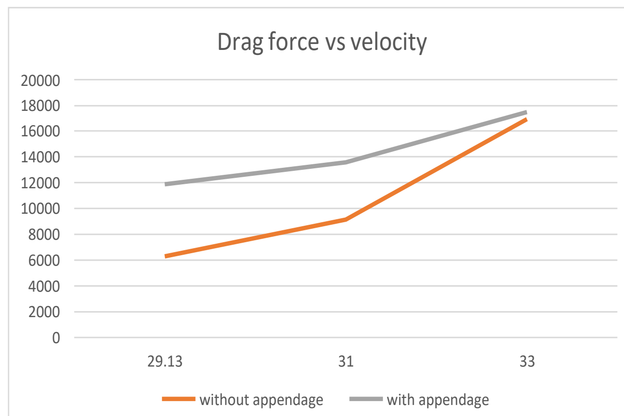


Fig.7

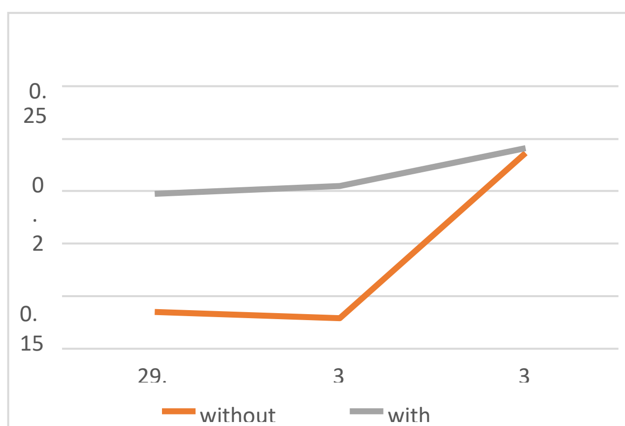
26.Velocity Streamline of Triangular Appendage:



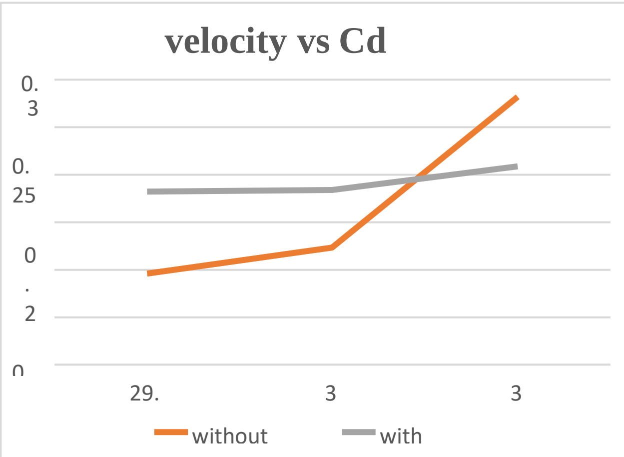
27. Drag force vs velocity:



28. Viscous drag vs velocity:



29. Coefficient of drag vs velocity graph:



30. Conclusion:

For cavitating flow over the hydrofoil, computational fluid dynamics has been performed using the FLUENT algorithm (NACA 0015). The boiling point of a liquidis when cavitation begins, and how quickly it collapses depends on the velocity. The fluid flow's velocity has an impact on the bubble's size. Three different cavitation numbers and velocities were simulated. The bubble develops and bursts on the surface when the ship-mounted hydrofoil glides slowly beneath the water. It generates excessive drag, vibration, and noise. Hyper cavitation is only helpful for the maritime sector when it occurs at high speeds with low cavitation numbers. Minimizing cavitation's impacts is a difficult task. A novel technique was used to lessen the impact of cavitation. To enhance the volume percentage of vapour and decrease collapse rate, change the design of the hydrofoil and add sharp comer (APPENDAGE) on both sides. The main benefit is shrinking the bubble size while maintaining almost the same amount of drag as the result produced without the appendage.

REFERENCES:

[1] Kjeldsen M, Arnd: REA, Effertz M (2000), "Characteristics of Sheet/Cloud Cavitation". J Fluids Eng 122:481-487.

[2] Rowe, A. and Blottiaux, O. (1993), "Aspects of Modeling Partially Cavitating Flows". J. Ship Research, v37:39-50.

[3] Amromin, E.L. (2007), "Determination of Cavity Detachment for Sheet Cavitation". J. Fluids Eng, v129:1105-1111.

[4] Rhic, C.M and Chow, W.L. (1983), "A Numerical Study of the Turbulent Flow past an Isolated Airfoil with Trailing Edge Separation" ALAA Journal 21 (1983):1525-1532.

[5] Yongliang Chen, Heister. S.D (1996). "Modeling Hydrodynamic Non-equilibrium in Cavitating Flows" Journal of Fluids Engineering v118:172-178.

[6] J.N. Newman (1978), "Marine Hydrodynamics", MIT.

[7] C. E. Brennen (1995), "Cavitation and Bubble Dynamics", Oxford University Press.

[8] Cartellier and J. L.Achard (1991), "Local Phase Detection Probes in Fluid/Fluid Two Phase Flows" Rev. Sci. Instrum. 62(2).

[9] S. F. Jones, G. M. Evans and K. P. Galvin (1999), "Bubble Nucleation from Gas Cavities", Advances in Colloid and Interface science, 80 27-50.

[10] Avellen, F., Henry, P. Ryhming, LL. (1987), "A New High Speed Cavitation Tunnel" ASME Winter Annual Meeting Boston 57-49-60.