Study on dependence of Cavitation number on Supercavitation phenomenon using Transient Multiphase Analysis

Siddhant Vikram Avhad

Student Mechanical Engineering, Sinhgad College of Engineering, Pune, India

Abstract: Cavitation causes many adverse effects in engineering applications, but this phenomenon could be used in a different way to reduce the viscous drag on objects travelling under water. As water due to its high density will impose high drag force. With the help of cavitation bubble the drag could be reduced drastically as the body of the object will be covered with less dense gas or vapour. Thus, it becomes important to study the cavitation parameters to predict the shape of cavity formed and drag force experienced by the object to ensure efficient travel under water. Cavitation number is the key factor affecting the cavitation phenomenon thus it is the independent quantity during analysis. Before running the simulation, grid independence test was performed to enhance the accuracy and reduce errors due to meshing. The article gives insight about the setting up of transient multiphase simulation for supercavitating flows. The time-stepping method selected for the simulation was "Variable" which ensures adaptive time-step size according to the mesh without changing the courant number in any case. The dependence of surface closure time on cavitation number is studied by observing the vapor volume fraction contour plots. Further the graphs of cavity diameter and drag coefficient are plotted considering cavitation number with less than 11% of error as compared to empirical formula stated before. Thus, using this simulation model with similar setup cavitation phenomenon could be studied reducing the initial errors faced due to meshing and improper setup of the model.

Index Terms: Supercavitation, Disk-Cavitators, Multiphase analysis, Drag Coefficient, Cavitation number.

I. INTRODUCTION

Cavitation phenomenon is known widely for its disadvantages such as erosion of Turbine blades, Marine propellers, Centrifugal pumps etc. The cause of this cavitation is the Pressure drop in some areas due to high velocity of fluid. Thus, there is distortion of flow and cavity bubbles are formed in the areas where the pressure is below the vapor pressure of water and the cavity collapses when it reaches the higher-pressure region. This phenomenon of Cavitation reduces the contact area of the body with the fluid thereby reducing the skin friction drag. Various simulation works have been done before on Supercavitating bodies studying cavity formation stages for different nose shapes [1]. Also, practical experimentation had been performed in other works using measurement techniques to record projectile speed, supercavity dimensions and target impact location [2]. In other literatures experiments had been performed in water tunnel to investigate the supercavity closure mechanisms with an objective to understand the physical mechanisms determining closure formation [3]. The shape of the cavity formed plays a great role as it is this cavity which avoids the viscous drag of water. The shape of the cavity is investigated experimentally by studying the cavity development over bodies in duct flow[4]. The drag reduction capacity of supercavitation is evaluated by comparing with a moving vehicle launching at the same speed but without supercavitation and the results show that the supercavitation reduces the drag of the vehicle dramatically [5]. Also, literatures state that semi-empirical theory and the potential theory could give an accurate result quickly, however they are not so versatile for the transient flow and complex geometric configuration. Meanwhile they are incapable to give detailed flow behavior of the cavitation [5]. Thus, it becomes very important to study proper implementation of the CFD model to simulate transient behavior of supercavitating flow. Numerically solving using CFD approach one can test multiple geometries without using the cavitation tunnels more easily.

In this article we have focused on the dependency of the drag force and shape of the cavity formed on cavitation number by solving the Transient multiphase model. Firstly, the CAD model of the geometry was prepared and after completing the mesh quality checks the mesh setting which gave minimal error was considered for the simulation. Further the model was imported into Fluent and solved using "VOF" and "K-Epsilon turbulence model". The graphs of cavity diameter and drag coefficient are plotted versus the cavitation number. Further a relation is derived for drag coefficient in terms of cavitation number and compared with the empirical formula from previous literature to find the absolute error.

II. CAD MODELING AND MESHING

A 2-d axisymmetric model of the supercavitating body was used for simulation as shown in Figure 1 which was loosely based on the design of Russian Supercavitating torpedo, 'The Shkval'. The length to diameter ratio of the body was taken like that of the Torpedo i.e. equal to 15 [6]. Although it is the diameter of the disk-cavitator which decides the cavity shape and the drag values but there was need to set the values of other dimensions too, thus the dimensions were fetched from the actual application of the phenomenon.

Figure 1. CAD model used for CFD analysis



Meshing of the model was done keeping in mind the flow area to be monitored the mesh was refined in those areas as shown in Figure 2 also inflation layers were added on entire surface of the body as the Near Wall settings were set to "Enhanced Wall Treatment" thus a fine mesh would be required near the wall. Maximum number of inflation layers added were 5 with first layer height of 1mm and growth rate equal to 1.2.





Grid independence test was performed for the model to minimize the errors due to meshing. The mesh was made denser by reducing the mesh element size by 1.5 times of initial value. The parameters monitored for the study were drag coefficient and vapour volume fraction. Table 1 depicts the values of output parameters for different mesh sizes:

Edge sizing on body	Mesh Element	Number of nodes	1 Mesh Element Count	Drag Coefficient	Vapour Volume Fraction
	Count				
8mm	10530	10786	9.49668e-05	0.993	0.0714624
6mm	21021	21377	4.75715e-05	0.965	0.0770533
4mm	42215	42667	2.36883e-05	0.983	0.0919348
3.5mm	53957	54439	1.85333e-05	0.938	0.0830123
3mm	77644	78154	1.28793e-05	0.919	0.0843943

Table 1. Values of Grid independence test Parameters

The graphs for above mentioned Output parameters were plotted using "1/Mesh element count" as the independent quantity as shown in Figure 3 and Figure 4. From this data a linear regression fit line was plotted for each graph which gave its intercept value

as the value corresponding to the finest mesh size. This value was the grid independent value for the Output Parameters. Further this value was used to calculate Percentage Error using Equation (2.1) for each of the Element Set used for grid Independence test as mentioned in Table 2.

 $Percentage \ Error(\%) = \frac{Output \ Parameter \ Value - Grid \ Independent \ Parameter \ Value}{Grid \ Independent \ Parameter \ Value} \times 100$ (2.1)



Figure 3. Drag Coefficient Mesh Independence Plot





Table 2. Percentage error in Grid independence test Parameters

1	Percentage Error (%)	
Mesh Element Count		
	Drag Coefficient	Vapour Volume Fraction
9.49668e-05	6.43	-19.80
4.75715e-05	3.43	-13.53
2.36883e-05	5.36	3.17
1.85333e-05	0.53	-6.84
1.28793e-05	-1.5	-5.29

The mesh with 53957 number of elements was finalized for the further study as it was the best compromise between the Computational time and the accuracy of the solution.

III. SETUP OF MODEL IN FLUENT

The operating temperature was taken to be 20° Celsius. The vapour pressure of water at this temperature was found to be equal to 2643.383Pa. Thus, the cavitation pressure input value was given equal to the vapour pressure of water at that temperature i.e. 2643.383Pa. The VOF model was used with number of Eulerian Phases equal to '2' i.e. 'water' and 'vapor' to model Cavitation

using Realizable K-Epsilon viscous model which is suitable for complex shear flows. Also, the Turbulence setting for Near-Wall Treatment was set to "Enhanced Wall Treatment" because it is a y+ intensive method and will act like a Wall function if the first grid point is in the log-layer. In the Solution Methods panel the scheme was set to SIMPLE, with Spatial-Discretization as 'Least square cell based' for Gradient, 'PRESTO!' for Pressure and as the velocity range was way too higher for all the iterations performed 'Second Order Upwind' was used for Momentum, Turbulent Kinetic Energy, Turbulent Dissipation Rate and Energy.

As shown in Figure 5, the left edge of the domain was selected as inlet and with specified velocities corresponding to the Cavitation number ranging from 0.02 to 0.9 as shown in Table 3. The Right edge of the domain was set as Outlet with Gauge Pressure equal to zero. The lower edge was set as the axis of symmetry. The upper edge of domain was specified as Symmetry to avoid boundary layer formation near that edge which would increase the computational time.



Figure 5. Geometry of model specifying boundary conditions.

Table 3. Different cases considered for Study

Cases	Cavitation Number	Velocity
Case 1	0.020549403	98.09
Case 2	0.029994637	81.19
Case 3	0.04380954	67.18
Case 4	0.063912634	55.62
Case 5	0.093358843	46.02

Time step size

The Time stepping method was selected to be 'Variable'. In this method the Timestep size changes according to the mesh size in domain with respect to Equation (3.2). The Global courant number value was set to 0.25 as given in Equation (3.1).

Global Courant Number =
$$\frac{u\Delta t}{\Delta x} = 0.25$$
 (3.1)

$$\Delta t = \frac{0.25 \times \Delta x}{u} \tag{3.2}$$

Where 'u' is the fluid speed, Δt is the Timestep size and Δx is the mesh size. Thus, according to the above equation, the Minimum and Maximum Timestep sizes were calculated corresponding to the Minimum and Maximum mesh sizes present in the domain respectively. Thus, the timestep size would change within in the given limits.

The Timestep sizes were updated for each Case mentioned in Table 3 as all the cases were conducted at different velocities.

IV. RESULTS AND DISCUSSION

Time of Surface Closure The

Table 4 depicts Contours of Vapour Volume Fraction at different Cavitation numbers but at same end time equal to 0.07sec.

Cavitation Number	Contours of vapour volume Fraction	
0.020549403	Upport Volume Fraction ANS 1,000+100 8,1678-001 8,1678-001 6,676-001 6,5038-001 6,676-001 5,000-003 -400 (4 1 4 2 4 4 4 2 4 4 1 4 4 4 4	5
0.029994637	Vigour Volume Fraction ANS 1.000+000 9.107-601 9.607-601 9.607-601 9.500-601 9.500-601 9.500-601 9.500-601 9.333-601 9.333-601 9.333-602 9.000+000	5
0.04380954	Uppour Volume Fraction ANS 8.1676-001 8.1676-001 9.1676-001 6.676-001 5.6338-001 6.8338-001 2.8006-001 6.8338-001 3.3338-001 8.338-001 2.8006-001 8.338-001 3.338-001 8.338-001 2.8006-001 6.8338-001 3.338-001 2.8006-001 0.0006+000 0.0006+000	S a
0.063912634	ANS 4.000+000 4.100+001 4.107+001 5.000+001 5.000+001 4.107+001 5.000+000 5.000+000+00+000+000+00+00+00+00+00+00+00	Si
0.093358843	Vacour Volume Fraction V111000+000 1 1572-001 3 333-001 5 503-001 5 503-001 3 333-001 2 2000-001 8 333-001 2 300-001 8 333-001 2 300-001 8 333-001 8 333-001 8 333-001 8 333-001	5 .0

Table 4. Plots of Vapour Volume Fraction Contour at t = 0.07seconds

C T 7

It is clear from the

Table 4 that as the Cavitation Number increases The Cavity Length (1) and Diameter (d) decreases. Thus, it is evident that the cavity formation depends on the velocity of the body. The time required for enveloping whole body will be different for different inlet velocities. A relation between the Fluid Free Stream Velocity and Time has been taken from the literatures published before. The following Equation (4.1) states that the dimensionless time of surface closure is roughly constant [7]:

$$T = \frac{tu}{d} \tag{4.1}$$

Where T = Dimensionless time of surface closure of water entry, t = Time after water impact (Flow time), u = Free Stream Velocity, d = Diameter of the Disk Cavitator. Following

Table 5 depicts the Vapour Volume Fraction Contours for different cases at flow time corresponding to the time required for full surface closure and the values of 'T' according to the Equation (4.1).

713

t	u	Т	Contours of Vapour Volume Fraction	
0.015386	98.09	30.18	Vepour. Volume Fraction 1 000e+000 0.167e-001 8.338-001 8.00e+001 8.00e+001 8.00e+001 1.00e+001 1.00e+001 1.00e+001 1.00e+001 0.00e+000 0.00e+000	
0.0186668	81.19	30.31	Vegour Virkame Fraction ANSYS 1 0.00e+0.00 1 1 3.334-001 2.500e+0.00 1 8.853e+0.01 8.853e+0.01 8 8.853e+0.01 8.853e+0.01 8 8.00e+0.01 1 1 1.67e+0.01 8.853e+0.01 1 9.80e+0.01 1 1 8.858e+0.01 1 1 9.80e+0.01 1	
0.0232273	67.18	31.20	Vegour Virlame Fraction ANSYS 1 1.000e+000 1 1 3.33x+001 1 2.500e+001 8.853x+001 1 8.353x+001 1 1 1.107e+001 8.000e+001 1 8.353x+001 1 1 1.107e+001 1 1 1.107e+000 1 1	
0.0308215	55.62	34.28	Viscus Volume Fraction ANSYS 1.000+000 1.000+000 1.1000+001 1.000+001 1.338-001 1.000+001 1.500+001 1.500+001 1.500+001 1.532+001 1.500+001 1.532+001 1.532+001 1.552+001 1.552+001 1.552+001 1.552+001 1.552+001 1.552+001 1.552+001 1.552+000 1.552+001	
0.0332374	46.02	30.59	Vogou Volum Franton VY1 1000+001 8 107x-001 8 830x-001 6 000x-001 8 830x-001 6 000x-001 8 333x-001 1 607x-001 8 835x-002 1 000x-000	

Table 5. Comparison of Contours showing full surface closure of body and their corresponding Dimensionless time for surface closure.

From the

Table 5 the Dimensionless time of surface closure of water entry (T) was calculated and was found to be almost constant for all the cases thus validating the work.

Relation between Drag coefficient and Cavitation number

Table 6. Drag coefficient of the body at different Cavitation Numbers

Cavitation number	Drag Coefficient
0.020549403	0.936
0.029994637	0.938
0.04380954	0.932
0.063912634	0.97
0.093358843	0.924

The data in Table 6 was imported into MATLAB's Curve fitting toolbox and using Linear Regression, Equation (4.2) was derived which is as follows:

$$C_d = 0.933 \, (1 + \sigma) \tag{4.2}$$

Equation (4.2) was compared with the Empirical Equation (4.3) derived previously [7] which is as follows:

$$C_d = C_{do} \left(1 + \sigma \right) \tag{4.3}$$

Where $C_d = \text{Drag}$ coefficient, $C_{do} = \text{Drag}$ coefficient at zero cavitation number which is equal to 0.84 for Disks and $\sigma = \text{Cavitation}$ number. The error in the equation obtained from CFD was calculated by comparing the C_{do} values which came out to be 11%.

Figure 6 depicts the graph of variation in the Dimensionless cavity diameter with the changing cavitation number. Where the cavity diameter is non-dimensionalized by dividing it with cavitator diameter. It could be inferred from the data points that the cavity radius decreases with increase in cavitation number.

Figure 6. Dimensionless cavity diameter at different Cavitation number.



V. CONCLUSION

The cavity formation has been studied by simulating the model for different Cavitation numbers and a relation between the Time of Surface Closure for different cavitation numbers has been verified. The mesh used for the study has been analyzed thoroughly and the meshed model with minimum errors in output parameters had been selected for the study. The Supercavitation phenomenon has been studied using VOF model and Realizable K-Epsilon Viscous model. The contour plots give a clear insight about the dependence of Cavitation number on the Time required for Surface closure. From these plots further it becomes easy to conclude that the case with least cavitation number took least time for cavity formation. Also, the dependence of drag coefficient on cavitation number has been studied, it was found that the drag coefficient increases with increase in cavitation number. Further a graphical representation of data points for the Maximum Cavity diameter corresponding to the respective Cavitation number states that the Cavity diameter increases with the decrease in cavitation number. Thus, the whole study revolves around the main factor related to cavitation phenomenon which is the Cavitation number.

REFERENCES

- [1] Mansour, Mohsen. (2016). COMPARISON STUDY OF SUPERCAVITATION PHENOMENA ON DIFFERENT PROJECTILES SHAPES IN TRANSIENT FLOW BY CFD.
- [2] Cameron, P. J. K., Rogers, P. H., Doane, J. W., & Gifford, D. H. (2011). An Experiment for the Study of Free-Flying Supercavitating Projectiles. *Journal of Fluids Engineering*, 133(2), 259–284. https://doi.org/10.1115/1.4003560
- [3] Karn, A., Arndt, R. E. A., & Hong, J. (2016). An experimental investigation into supercavity closure mechanisms. *Journal of Fluid Mechanics*, 789, 259–284. <u>https://doi.org/10.1017/jfm.2015.680</u>
- [4] Arad Ludar, L., & Gany, A. (2020). Experimental Study of Supercavitation Bubble Development over Bodies in a Duct Flow. *Journal of Marine Science and Engineering*, 8(1), 28. https://doi.org/10.3390/jmse8010028
- [5] Yang, D., Xiong, Y. L., & Guo, X. F. (2017). Drag reduction of a rapid vehicle in supercavitating flow. *International Journal of Naval Architecture and Ocean Engineering*, 9(1), 35–44. <u>https://doi.org/10.1016/j.ijnaoe.2016.07.003</u>
- [6] http://www.militaryperiscope.com/mdb-smpl/weapons/minetorp/torpedo/w0004768.shtml
- [7] DOI: 10.1115/1.4003560May, A., & NAVSEA HYDROBALLISTICS ADVISORY COMMITTEE SILVER SPRING MD. (1975). Water Entry and the Cavity-Running Behavior of Missiles. Defense Technical Information Center.