

# DESIGN OPTIMIZATION OF A CAVITATING SUBMERGED BODY USING COMPUTATIONAL FLUID DYNAMICS

M Sohaib<sup>1</sup>, M Ayub<sup>2</sup>, M. Rafique<sup>3</sup>, and M. A.Khan<sup>4</sup>  
National Engineering and Scientific Commission Islamabad Pakistan

## ABSTRACT

*Transient's analysis is performed to determine the cavitation inception on a submerged body and flow dynamics under different working conditions. Cavitation analysis is done for a particular geometry to investigate the location and determine the critical conditions for different depths at which cavitation take place. Simulation has been done using the commercial CFD code Fluent 6.2.16. Multiphase Mixture model and Standard K-  $\epsilon$  turbulence model with standard wall function is used in the study. Analysis determines the region and critical velocity for a particular depth at which cavitation occurs. The time dependent analysis provides detailed insight into the hydrodynamics and highlights the capabilities and limitations of the cavitation model used.*

## 1. INTRODUCTION

Applications of computational fluid dynamics (CFD) to the maritime industry continue to grow as this advanced technology takes advantage of the increasing speed of computers. Numerical approaches have evolved to a level of accuracy, which allows them to be used during the design process to estimate the forces and moments acting on the body during steady and unsteady motion and to predict the maneuvering performance of the vehicle moving underwater provided that have been validated to a certain level. Significant progress has been made in predicting flow characteristics around a given ship or marine hull. The marine designers can use this information to improve the submarine and ship's design. Prediction of drag on a ship hull is always a challenging task for a naval architect. At the start of the design process, hull forms are developed given certain requirements. One of the major design tasks is to estimate the powering performance so that propulsion requirements can be determined. Early estimates of resistance and power are often based on simple empirical formulas derived from data for similar ships. As the design process proceeds, a more reliable approach becomes necessary to predict resistance; scale-model testing has been generally adopted for this purpose. Cavitation plays an important role in many practical life problems. This phenomenon plays a significant role where high speed flows are involved. Its role in marine hydrodynamics is very important. While designing underwater moving vehicles, special attention is paid to it.

Cavitation in liquids takes place when the local pressure of liquid at a certain location is less than its vapor pressure. Since pressure in liquids is related to velocity, so at the regions where velocity is high enough so that the local pressure at that particular location becomes less than the vapor pressure, cavitation takes place.

Many problems related to structural strength of the objects are caused due to cavitation. Cavitation bubbles formed over the surface of underwater moving vehicles, cause the sever structural problems when these bubbles rupture. When these bubbles collapse, an enormous vibrations are produced which not only cause danger to structural strength of the object but also can cause many sever metallurgical problems. Role of cavitations in control system design is also matter of concern. Cavitation developed over a large scale result in decrease in drag which disturbs the control parameters. The decrease in drag is because of the reduced wetted area due to cavitation.

Cavitation produces enormous noise. This is a severe problem for strategic weapons. Submarines are designed to attack on enemy which could be possible if they are not detected by the enemy. Noise produced by cavitation may put their survival in danger because it act as an alarm to the enemy. Therefore cavitation in strategic weapons is strictly avoided. Cavitation could also used to design the high speed underwater moving vehicles. Supercavitating bodies have the cavitation bubble all around them, which result in reduced wetted area and hence the drag is reduced drastically.

Under Cavitation conditions, water is one of the worst actors regarding equipment damage due to cavitation. Water is hard on equipment in cavitation conditions for at least two reasons: relatively high density (small molecular size and heavy molecular weight), and a sharp well defined phase change behavior. The combination of small heavy molecules and high cavity wall implosion velocities (resulting from sharp and fast rate of phase change), results in the release of extreme inertial energies as the walls of cavity strike against each other and against objects in the fluid flow path during the cavity implosion. Liquids with molecules larger and more complex than water and non homogenous liquids such as many petrochemicals, cab be less much harmful to equipment in cavitation conditions because they often have a lower than water, their larger more complex molecular structure and sometimes non-homogenous nature causes blurred or less well defind phase change behavior, both of which reduce the rate of cavity creation and collapse, and the amount of energy released in the implosion. Therefore the amount of damage caused by cavition is reduced.

In the design process of a wide variety of fluid machinery, as the naval profile under investigations, the occurrence of cavitation is on of the most important aspects that need to be considered. Cavitation is the formation of vapor in a liquid in regions where liquid velocity is high and consequently pressure is low, i.e. becomes lower than the vapor pressure. The inception, development, and collapse of cavitation produce noise, vibration and even damage the solid wall.

Cavitation phenomena are of interest for a wide range of engineering fields, from fuel injection systems for internal combustion engines to hydraulic turbines and pumps of all size. In injection nozzles, cavitation has a strong influence on spray formation and atomization. In hydraulic machine the cavitation may occur along stationary parts and on the moving blades. Liquid fuel and oxidizer turbo pumps for rocket engines are usually operated under cavitating conditions with quite high rotational speed to attain high performance with their minimum size and weight. Cavitation of marine propellers may cause many problems, such as vibration noise and erosion on the blades. Marine propellers researchers and designers have made numerous efforts to reduce the effect of cavitation. However with recent high speed shallow draft ships, it is difficult to avoid cavitation without compromising the efficiency of propellers.

When a marine propeller is operated in the wake of a ship, the angle of attack of the incoming flow relative to the blade varies as the propeller rotates. This causes the periodic growth and collapse of cavity, and this fluctuating pattern is strongly related to the vibration and noise often associated with the cavitation. Propeller designer must therefore control the influence of cavitation rather than try to suppress its occurrence. As a result, the accurate prediction of cavitation is becoming increasingly important. Model tests provide valuable insights into the cavitation physics in various predetermined conditions, but they cost a significant amount of money and are vulnerable to slight flow condition changes inside cavitation tunnels. Different efforts have been given in literature to simulate cavitating flow with computational fluid dynamics (CFD) methods. However validation of computational results with experimental data is very fragmentary in many papers. The main cause is the lack of experimental data. In general the results are very promising. Several aspects of cavitation phenomena, of fundamental importance for the profiles design, can be accurately captured with the developed cavitation models.

## 2. GEOMETRY

Geometry used here is the standard hull models of submarine DARP 2 and the salient features of the geometry is listed here and sketched in the Fig. 1,2,&3. The detail of the geometry is as follows.

### Model

Length,  $L = 4.355$  m

Cylinder body Radius,  $R = 0.2539884$  m

Exit Diameter,  $D(\text{exit}) = 0.0075$  m

## 3. GRID GENERATION

For 2D simulation, structured grids are modeled for the analysis of submarine hull. Structured grids were modeled for the cavitation analysis at different working conditions. Three different Grid Models were constructed for hull body configuration having different number of blocks 4, 5 & 6 respectively. Grid independence was insured for three grids, grid1, grid2 & grid3.

For grid independence study, Coefficient of pressure was calculated and plotted for all the three grids shown in Fig 2. It was observed clearly that pressure coefficient results for all the three grids of different dimensions are very close to each other. Hence our analysis is independent of Grid model. Grid 2 having less dimensions is selected for the study to save computational time. This Grid model for the hull configurations consist of 5 blocks and after grid independence study this grid was selected for analysis. 2D axis-symmetric body was modeled to perform the cavitation analysis for the hull configuration at different conditions of pressure and velocity. Grid 2 having dimensions 60 x 80 points, in the Block-1 containing the Nose section, the Block-2 consists of a cylinder body model with the grid size of 40 x 80 points, the Block-3 is over the tail to the far field of the size of 60 x 80 points, the Block-4 is over the lower tail portion having size 30 x 80 points,

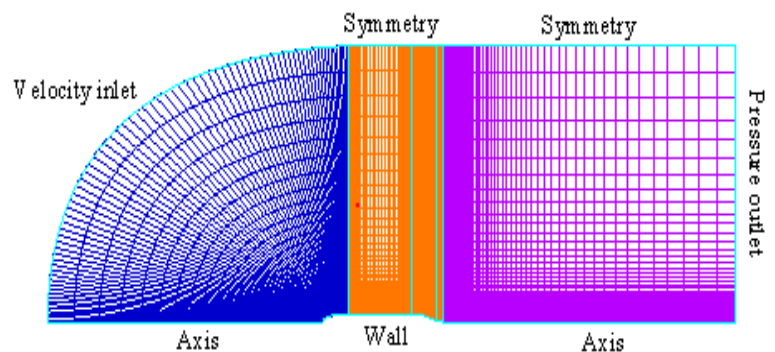


Fig.1 Grid 2 2-D axis symmetric Structured grid for Hull model

the Block-5 is between the body and last domain wall. This is a wake domain consisting of 60X80 points. Grid model used in the study was made by using PAKGRID [6]. Grid is shown in Fig1.

#### **4. FLOW CONDITIONS**

Flow conditions for the submarine DARPA 2 are reported as follows:

Free stream velocity = 15, 30, 45 m/s  
Reynolds number = 3.89E07,  
Angle of attack = 0 deg  
Pressure outlet = 2, 4, 5 bar  
Turbulence model = K- $\epsilon$  model  
Depth in water = 10, 20, 30, 40 m  
Near wall treatment = Standard Wall function  
Discretization scheme = First order upwind scheme

#### **5. BOUNDARY CONDITIONS**

Velocity Inlet conditions were imposed on the far field domains, while the pressure outlet condition was imposed at the exit of the submarine assembly. At the submarine walls, no-slip adiabatic Wall condition was imposed. For the interfaces between the different blocks, the contiguous interface boundary condition was used. 3-Dimensional half-body plane was taken as symmetry plane condition and Axis condition was applied on the nose and tail centerline.

#### **6. RESULTS & DISCUSSIONS**

The main purpose of the study is to analyze the hull geometry for cavitation. Emphasis has been given to the determine the cavitation existence and to locate the region where cavitation has occurred. Transient analysis is performed to understand how the cavity size grows with the time. Analysis is performed at different working conditions. In the first part of analysis velocity is kept constant while the underwater depth of the hull is set at 10m, 20 m, 30m, and 40m. The analysis is performed at fixed velocity of 30m/s while the pressure at three different depth locations were 2 bar, 4 bar and 5bar respectively.

At  $v = 30\text{m/s}$  &  $p = 3\text{bar}$ , static pressure distribution over the hull body is shown in fig 6. Pressure distribution over the body shows the regions where the low pressure is observed. Regions where the edges are sharp, pressure reduces and reduces beyond the vapor pressure. This phenomenon is clearly observed in fig 5 where volume fraction of water vapors is observed, showing the cavity on the nose region and on the tail region. These are the regions in the xy-plot of pressure, where the local pressure is less than the vapor pressure. At  $v = 30\text{m/s}$  &  $p = 4\text{bar}$ , In this case body being at greater depth 30m, causing increase in pressure. Volume faction of water vapors is reduced than at depth 10m. But still there are regions of low pressure causing cavitation. The size of the cavity is reduced shown in fig 7 & 8, this is because of the fact that as the pressure rises at greater depth and region where the pressure goes below the vapor pressure also reduces. Although the pressure reduces near the nose and tail regions but for the pressure to go below the vapor pressure from the high pressure is relatively less.

At  $v = 30\text{m/s}$  &  $p = 5\text{bar}$ , same phenomenon happens. Size of the cavity is further reduced at same locations, this is because of the fact that  $\frac{1}{\rho} \frac{dp}{dt}$  reduces at these locations but this reduction in pressure from such a high pressure is relatively less, shown in fig 9 & fig10.

In the second part of study, Pressure is kept constant and analysis is performed at constant depth of 10m and for three different velocity conditions, i.e.  $v = 15\text{ m/s}$ ,  $30\text{ m/s}$ ,  $45\text{ m/s}$ . At  $P = 2\text{bar}$ ,  $v = 15\text{m/s}$ ,  $t=10\text{ s}$ , in these conditions although the velocity is not that high but still the velocity at nose and tail regions is high enough. Since water being incompressible, the pressure at these regions is low enough so that it goes below the vapor pressure. Therefore volume fraction of water vapors at these region are observed forming cavities. Although the size of cavities formed is not so large. This fact is shown in fig11 & fig 12.

At  $P = 2\text{ bar}$ ,  $v = 30\text{m/s}$ , analysis show that this velocity is high enough to cause very high velocities at nose and tail regions, hence resulting in very low pressure. Pressure here is again low enough to be lower than vapor pressure. In this case, low pressure region is large causing bigger size cavity shown in fig 13. The location of cavity is well demonstrated in the static pressure graph fig14. At  $P = 2\text{bar}$ ,  $v = 45\text{m/s}$ , this velocity is high enough to cause very high velocity at sharp edges. Water being incompressible, causing large low pressure regions at these locations. Therefore in this case region where cavity formed is much larger.

In the analysis it was also observed that cavity size formed is time dependent up to time it is growing but when once it is grown, then it becomes time independent.

Hence while designing the underwater moving vehicle; it is very important to design such a shape that incorporates minimum sharp changes in geometry, and hence the low pressure regions are avoided. Cavitation analysis should be performed on every change in geometry to avoid cavitation.

Validation of results was done indirectly. Coefficient of pressure was calculated for Hull model using fluent and was compared with experimental results and results available in literature. It was observed clearly that since our results agree well with experimental results, therefore our CFD analysis is validated for pressure distribution over the Hull body. Now by comparing the pressure distribution over the Hull body for different working conditions, demonstrates that the cavities at the particular location of the low pressure regions are formed and the size of cavity depends upon the size of low pressure region.

Fig 2 Comparison of Cp-Distribution over the Hull body with Experiment

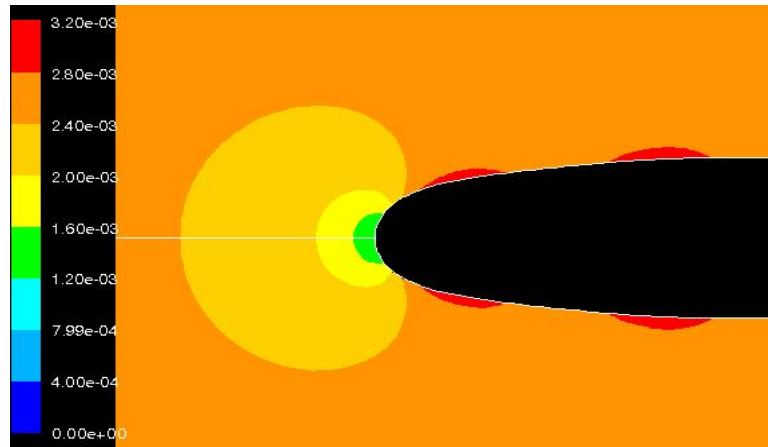
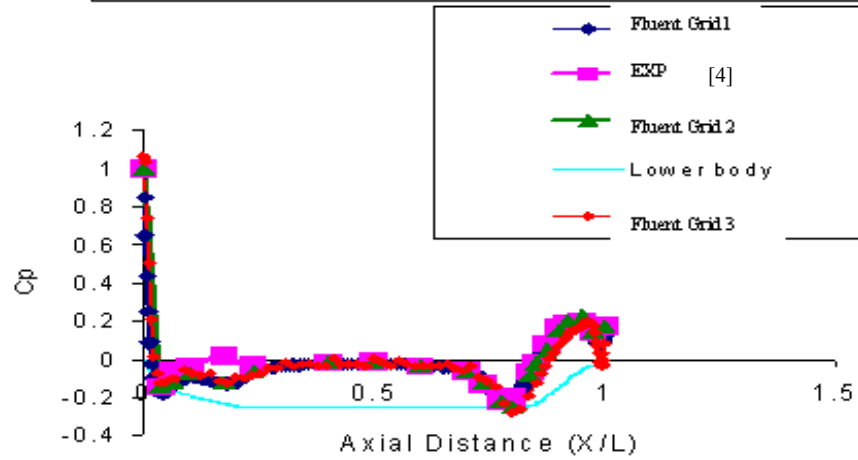


Fig 3 A typical bubble cavity formed on Nose of Hull model at  $P = 4$  bar,  $V = 30$  m/s,  $t = 20$  s

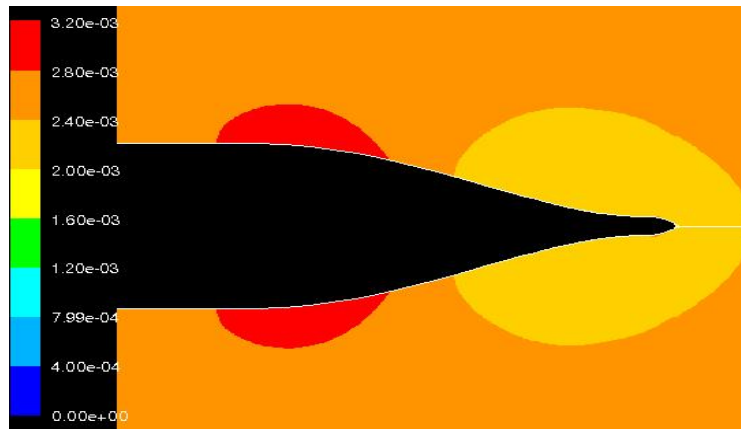


Fig 4 A typical bubble cavity formed on Tail of Hull model at  $P = 4$  bar,  $V = 30$  m/s,  $t = 20$  s

## **7. CONCLUSION**

- 1.** It is obvious from the analysis that cavitation takes place at the locations where flow encounters the sharp changes in geometry. At these locations local pressure drops below the vapor pressure, creating cavities there.
- 2.** It is also observed in the analysis that size of the cavity grows with the time until flow gets fully developed. When flow gets fully developed, region of low pressure is specified and cavities are formed at these specified regions forming specified size of the cavity. Outside this region pressure is not lower than vapor pressure, no cavity is observed.
- 3.** Analysis also shows that cavity formed remains intact with the wall at these conditions. For higher velocity and lower pressure, it may detach from the body.
- 4.** While designing, to rule out cavitation optimization should be done by analyzing the geometry at every condition to know limits of the designer.

## REFERENCES

- [1] Reboud, J. L., Delannoy, Y. (1994) "Two -Phase Flow Modeling of Unsteady Cavitation," Proceedings of 2<sup>nd</sup> International Symposium on Cavitation, Tokyo, Japan, pp. 39-44.
  
- [2] Saurel, R., Cocchi, J.P., Butler, P.B. (1999) "Numerical Study of Cavitation in the wake of a Hypervelocity Underwater Projectile," AIAA Journal of Propulsion and Power, Vol. 15, No. 4.
  
- [3] Reichardt, H. (1946) "The laws of Cavitation Bubbles at Axially Symmetrical Bodies in a flow," "Ministry of Aircraft Production Volkenrode ,MAP-VG, Reports and Translations 766, Office of Naval Research.
  
- [4] Cindy C Whitfield, Dr Eric Johnson, Dr. Fredrick Lutze "Steady and Unsteady Force and moments Data on a DARPA2 Submarine", PhD Thesis Virginia Polytechnic Institute and State University, Blacksburg, Virginia.
  
- [5] Lindau, J.W., Kunz, R.F., Gilbeling, H.J. (2000) "Validation of High Reynolds Number, Unsteady Multiphase CFD Modeling for Naval Applications," presented at the 23<sup>rd</sup> Symposium on Naval Hydrodynamics, Val de Reuil ,France



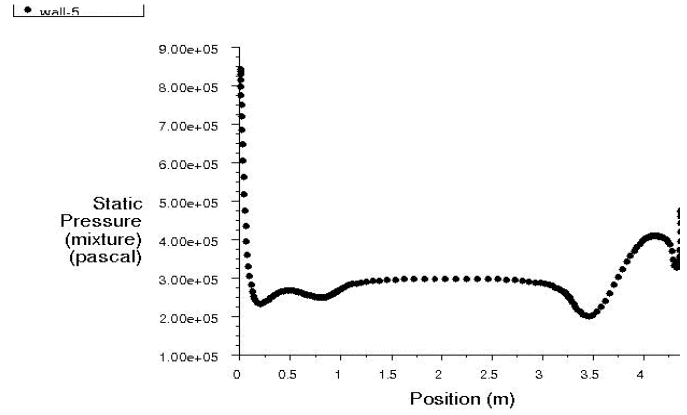
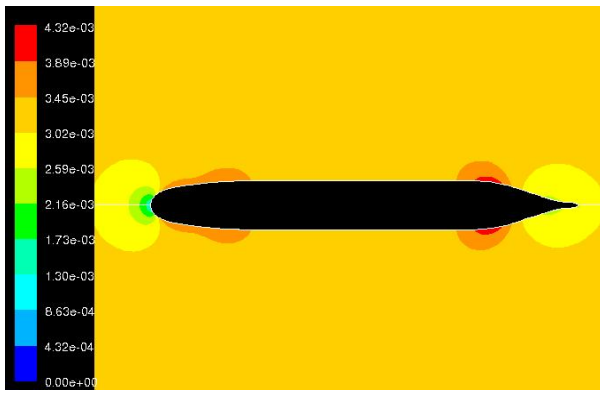


Fig 6 XY-plot of pressure distribution Over the Hull wall at P = 3 bar, V= 30m/s, t = 20s

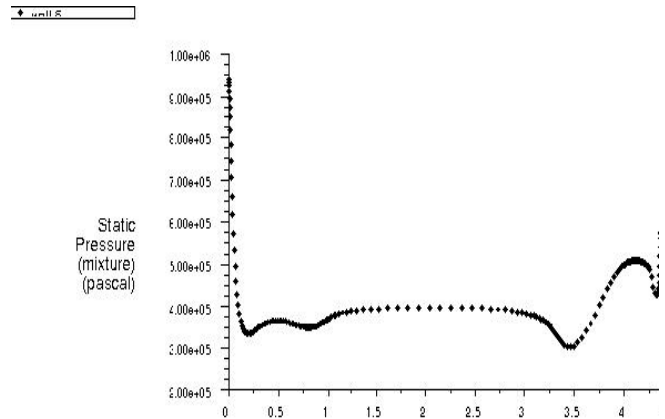
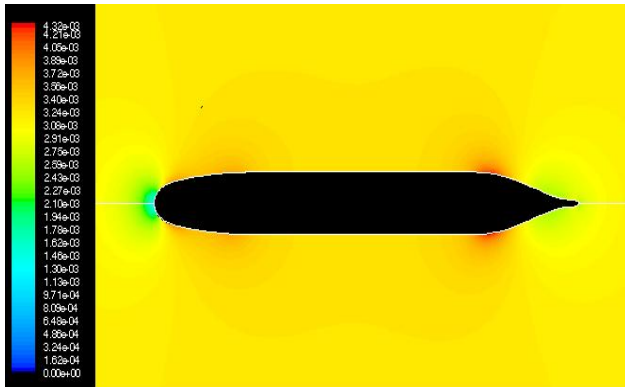


Fig 8 XY-plot of pressure distribution Over the Hull wall at P = 4 bar, V= 30m/s, t = 20s

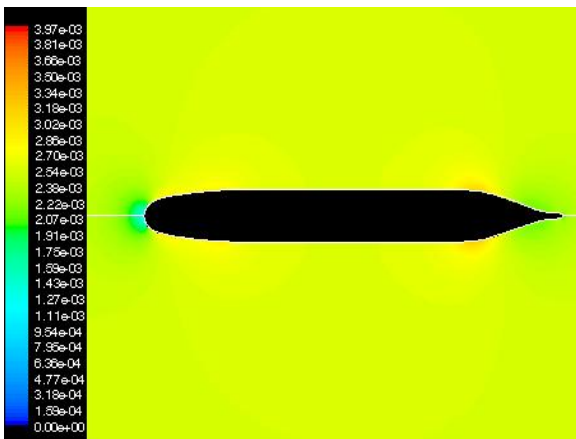


Fig 9 Volume fraction of water vapors using Mixture model at P = 5 bar, V= 30m/s, t = 20s

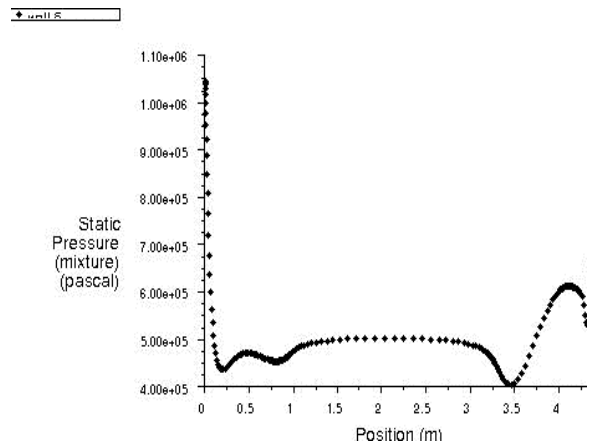


Fig 10 XY-plot of pressure distribution Over the Hull wall at P = 5 bar, V= 30m/s, t = 20s

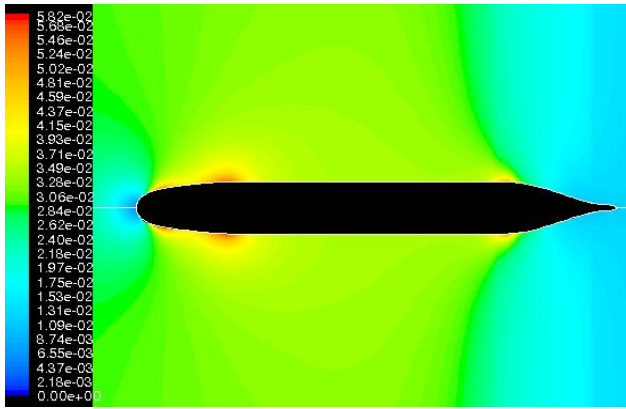


Fig 11 Volume fraction of water vapors using Mixture model at P = 2 bar, V= 15m/s, t = 10s

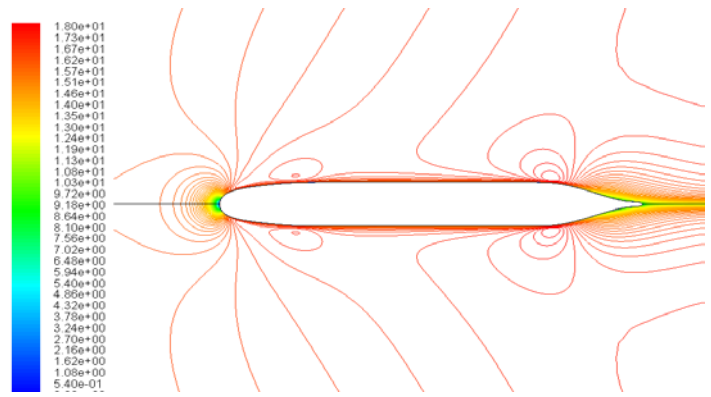


Fig 12 Velocity contours of water vapors using Mixture model at P = 2 bar, V= 15m/s, t = 10s

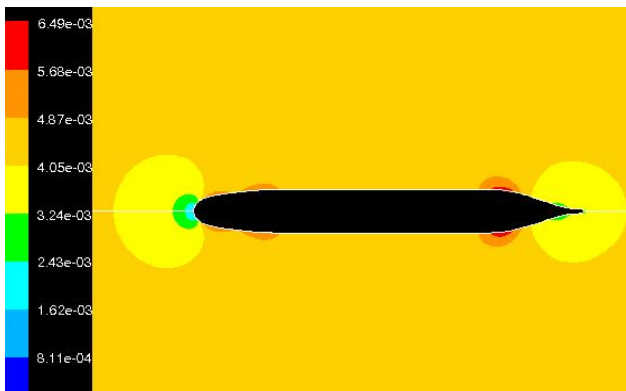


Fig 13 Volume fraction of water vapors using Mixture model at P = 2 bar, V= 30m/s, t = 20s

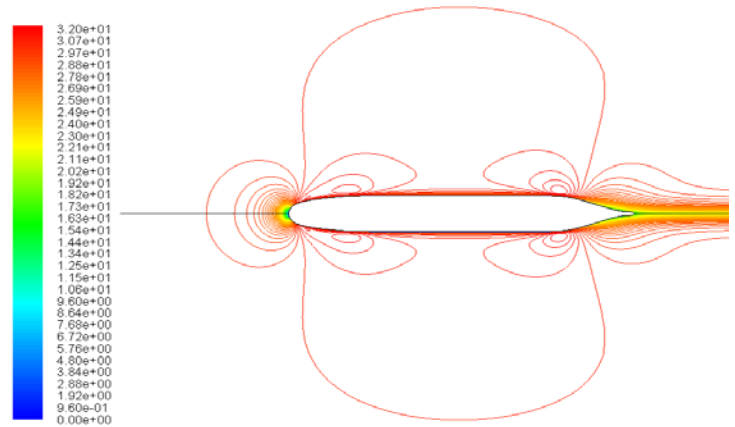


Fig 14 Velocity contours over the Hull wall using Mixture model at P = 2 bar, V= 30m/s, t = 20s



